



Anilam Electronics Corporation

CRUSADER SERIES-M

PROGRAMMING MANUAL

PART NO. 700-135

TABLE OF CONTENTS

<u>TITLE</u>	<u>PAGE No.</u>
Safety Notice	
Introduction	
SECTION 1: CNC ORIENTATION AND START UP	
A. Power Supply	
B. Turning the System On	
C. Using the Machine Manually	
SECTION 2: PART DIMENSIONING AND THEORY	
A. Axis Direction	
B. Absolute Dimensions	
C. Incremental Dimensions	
SECTION 3: VARIOUS MODES OF OPERATION	
A. Starting the Servo Motors	
B. Using Manual Mode	
C. Setting the Absolute Zero	
D. Program Enter	
E. Program Check	
F. Single Step	
G. Auto	
H. FEED %	
I. Inch/Millimeter	
J. Softkeys	
SECTION 4: PROGRAMMING KEYBOARD DEFINITIONS	
A. Display Position	
B. Do Loops (DO)	
C. Subroutines (SUBR)	
D. Call	
E. End	
F. Rapid	
G. Feed	
H. Absolute/Incremental (ABS/INC)	
I. Dwell	
J. Auxiliary (AUX)	
K. Tool (T)	
L. Arc Clockwise (ARC CW)	
M. Arc Counterclockwise (ARC CCW)	
N. G Codes	
O. Variables (V Codes)	
SECTION 5: THE SERIES-M CONTROL SCREEN	
SECTION 6: PREPARATION FOR PROGRAMMING	
A. Holding the Workpiece	
B. Determining a Reference Point	
C. Tool Selection	
D. Beginning a Program (One & Two Axis Linear Interpolation)	
E. Clearing the Memory	
F. Program Entry	

- G. Editing
- H. Program Execution
- I. Circular Motion

SECTION 7: PROGRAMMING EXAMPLE: DRILLING AND SPOT DRILLING

- A. Drilling and Boring Canned Cycles
- B. Drilling Programming Example
- C. Entering the Program into the Control
- D. Checking the Program
- E. Add Event
- F. Delete Event (p4)
- G. Machine Set-up
- H. Setting Tool Length Offsets
- I. Review of Programming Format
- J. Checking the Program with Graphics
- K. Running the Program

SECTION 8: MORE DRILLING EXAMPLES

- A. Drilling Format Rules
- B. Do Loops Programming Example
- C. Subroutines Programming Example
- D. Nesting Programming Example
- E. Peck Drilling Programming Example
- F. Mirror Image Programming Example

SECTION 9: MILLING EXAMPLES

- A. Circular Milling Programming Example
- B. Mirror Image Programming Example
- C. X and Z Contouring Example

SECTION 10: UNDERSTANDING CUTTER COMPENSATION

- A. Calculating the Tool Path
- B. Distance of Compensation
- C. Turning Compensation On and Off
- D. Compensating an Inside Pocket
- E. Cutter Compensation Programming Example
- F. Rules of Cutter Compensation
- G. Pocket Cutter Compensation Example

SECTION 11: UNDERSTANDING G-CODES AND CANNED CYCLES

- A. Defining G-codes
- B. Ellipse
- C. Planar or Helical Spirals
- D. Rapid Polar Moves with Spindle at Polar Center
- E. Feed Polar Moves with Spindle at Polar Center
- F. Rapid Polar Moves with Center Specified
- G. Feed Polar Moves with Center Specified
- H. G-codes Related to Cutter Compensation
- I. Mold Rotation
- J. Conical Cavity Milling Cycle
- K. Radial Conical Milling Cycle
- L. Polar Rotation
- M. Scaling
- N. Frame Milling
- O. Hole Milling Cycle

- P. Circular Pocket Milling
- Q. Rectangular Pocket Milling
- R. Bolt Hole Circles
- S. Helical Interpolation

SECTION 12: UNDERSTANDING THE HELP MENU

- A. When to Use the Help Menu
- B. G-code Help
- C. AUX-code Help
- D. RS-232 Help
- E. Arc Help

SECTION 13: AUXILIARY CODES

- A. AUX-code Listings
- B. Mirror Image
- C. Continuous Path
- D. Absolute Zero Shift
- E. Software Limits
- F. Backlash Compensation
- G. Feedrate Override
- H. Feed and Rapid Override
- I. Cancel Vector Rapid
- J. Set Vector Rapid
- K. Program Enter Lock
- L. Beeper Off/On
- M. Display Free Memory
- N. Single Step (Event)
- O. Single Step (Move)
- P. AUX 1000 Cancellation
- Q. Plane Rotation Cancellation
- R. Plane Rotation Set

SECTION 14: THE SERIES-M GRAPHICS FEATURE

- A. How to Use Graphics
- B. Zero Set
- C. Scaling
- D. Execute Blocks
- E. Cutter Compensation (On or Off)
- F. Speed
- G. Running Graphics

SECTION 15: MISCELLANEOUS FEATURES

- A. Tape Cassette Program Storage
- B. RS-232 Auxiliary Codes
- C. Error Code Messages
- D. Diagnostics

SECTION 16: ADDITIONAL DEMONSTRATION PROGRAMS

- A. Plane Rotation
- B. Elbow Cavity Milling
- C. Mold Rotation

SAFETY NOTICE

FOR CLARITY PURPOSES, THE DRAWINGS IN THIS MANUAL HAVE BEEN ILLUSTRATED WITHOUT PROTECTIVE GUARDS IN PLACE.

WHEN APPLYING THIS CONTROL IN A RETROFIT APPLICATION, YOU ARE CONVERTING A MANUALLY CONTROLLED MACHINE. CONSIDERATION MUST BE GIVEN TO MEETING APPLICABLE SAFETY AND GUARDING STANDARDS. THESE STANDARDS CAN BE OBTAINED FROM:

1. "Safety Requirements for the Construction, Care and Use of Drilling, Milling and Boring Machines", ANSI Standards Institute, 1430 Broadway, New York, New York 10018.
2. "Point of Operation Guarding" Section 1910.212, U.S. Occupational Safety and Health Administration.
3. "Power Transmission Guarding" Section 1910.219, U.S. Occupational Safety and Health Administration.

(Information in this manual is subject to change without notice.)

PART NO:
PRINTING DATE:

HELP MENU (A)

=====

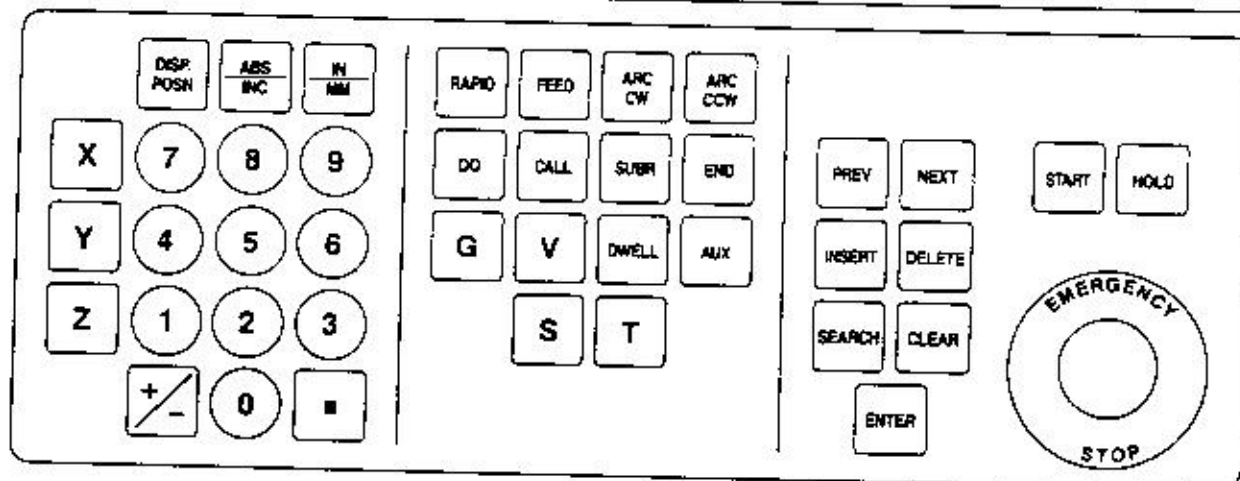
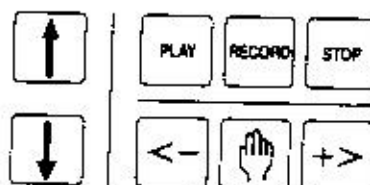
Push G for help G codes.
 Push AUX for help AUX functions.
 (B) Push RAPID for help RS232 setup. (C)
 Push ARC CW for help Arc Clockwise.
 Push ARC CCW for help Arc Counter-clockwise.
 Push T for help Tool Settings.

=====

(D)

END HELP

CRUSADER SERIES M MULTIPROCESSOR CONTROL SYSTEM





INTRODUCTION

This guide should be used by the machine operator to learn the programming format for the Crusader Series-M CNC Microprocessor Control system. It also will serve as a quick reference for CNC programmers that wish to familiarize themselves with the Series-M.

This manual is designed to teach. In order to use it effectively, the instructions and examples must be followed in step-by-step order. Understand each section before going on to the next section. This approach is necessary to thoroughly familiarize you with the Series-M and its functions.

This manual will first explain important points of axis direction and dimensioning theory. It is necessary to understand these subjects before using any CNC.

This manual will then define the functions of each button on the programming keyboard. References are also given so the operator may see each button in use within the many working examples later included in the manual.

As a student learning from this manual, you will be taken through the programming of a simple part. You will also dry run this part using the Crusader Series-M. Also, several "hands-on" programming examples are included to teach you about various Series-M features such as tool length offsets, mirror image, do loops and cutter diameter compensation, to name a few.

Finally, this manual explores the Series-M canned cycles in detail, showing you how the Series-M will greatly reduce your programming time. A review of the Table of Contents will give you a concise picture of the Series-M's capabilities.

Bear in mind that the Series-M is programmable and will follow all instructions exactly. If unsafe or incorrect instructions are given to the Series-M, it will follow these instructions.

One last note, ensure that you understand the use of the **Emergency Stop** button to avoid damage or injury. Do not hesitate to use Emergency Stop if a dangerous or unsafe condition arises.

SECTION 1
CNC ORIENTATION AND START UP

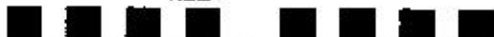
SECTION 1: CNC ORIENTATION AND START UP

A. POWER SUPPLY

Power is supplied to the system thru 115 volt (230 volt optional) power supply cable. This is attached to the servo amplifier cabinet usually located on the right side of the machine column as shown in figure 2. Power must be constant at a minimum of 105 volts AC and it must be properly grounded. The power cord can now be plugged into an AC power source.

B. TURN THE SYSTEM ON

To turn system on, the console must first be started. Do this by placing On-off switch located on back side of servo cabinet in the ON position. The CRT should display as shown:

1		Feedrate: 0-100%		Tool 0	
X	+000.0000	Manual		Posn Inch In-Posn	
Y	+000.0000				
Z	+000.0000				
Hold					
1					
2					
HELP					
					

Allow 30 seconds for this to happen. If it fails to come up as shown, turn ON/OFF switch off, wait for 1 minute and try again. If the unit still fails to come up, call your nearest Anilam representative or call the Anilam service hot line:

1-800-327-6340

C. USING THE MACHINE MANUALLY

On the back side of servo amp cabinet is the spindle toggle switch marked Manual/Auto. Put this in the manual position. This is the position the switch will be in when the machine is to be operated with the manual hand wheels without the use of the servo motors.

To start the spindle, put toggle switch in the manual position, then push in, twist, and pull out the Emergency Stop. Press the **RESET** button on the front of the servo amp cabinet, then turn the spindle motor ON with its original switch. The machine can now be used as a manual mill or for set up purposes. Use the machine in this mode while reading Section 2 (Part Dimensioning and Theory).

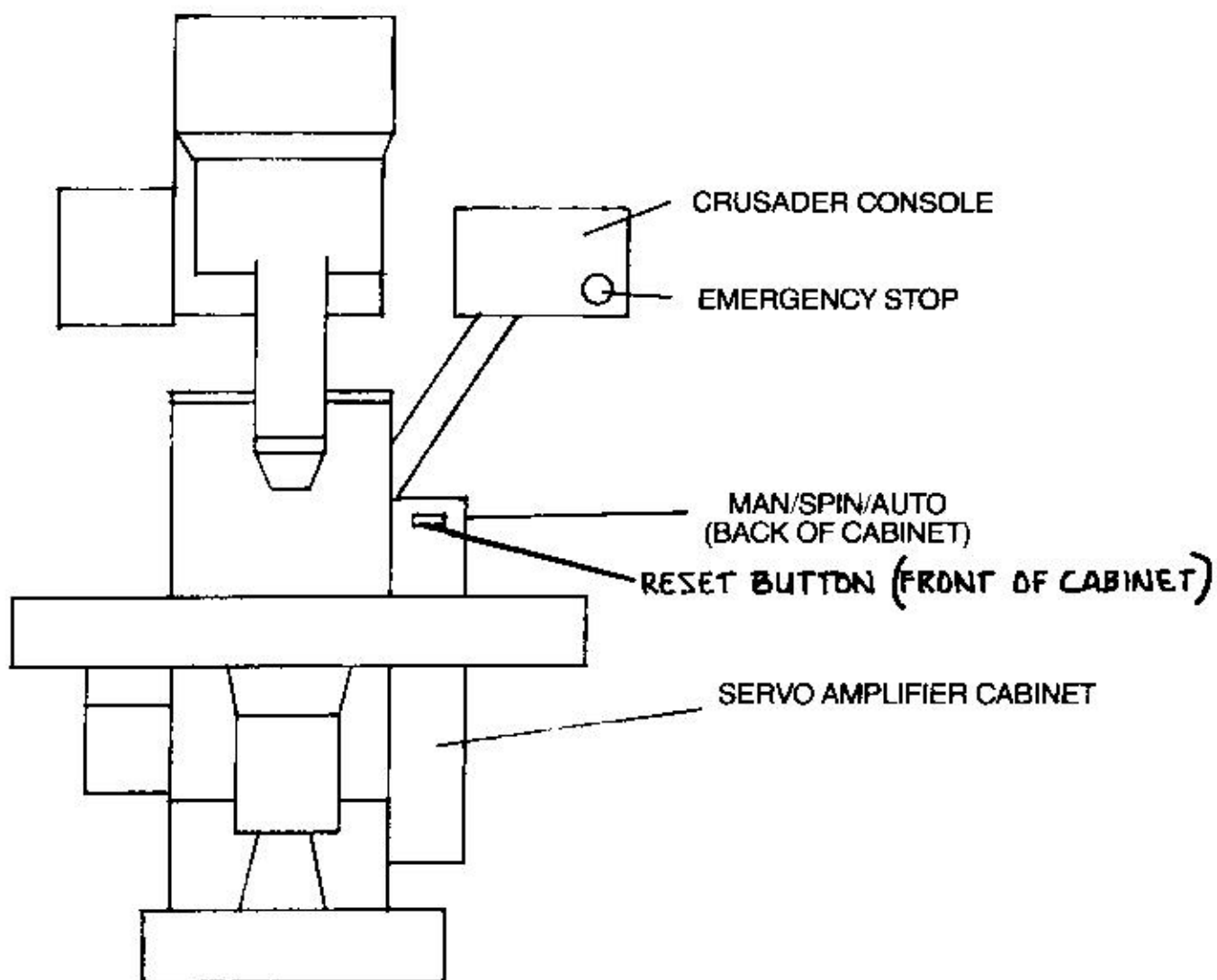


Figure 2

SECTION 2

PART DIMENSIONING AND THEORY

CHAPTER 2: PART DIMENSIONING AND THEORY:D

A. AXIS DIRECTION

Figure 3 shows a simple number line. It has a reference point zero, which is called absolute zero, and is divided into increments. Moving away from absolute zero to the right, are positive numbers. Moving from absolute zero to the left, are negative numbers. The plus or minus sign of a number indicates its relationship to absolute zero. Positive numbers have a (+) sign, and a negative numbers have (-) sign. To name a point 3 increments to the left of absolute zero, use the term -3.

Absolute.

Imagine a number line that is placed along each axis of the mill that we can control. The increments of the number line are .0005 (.01mm). An absolute zero can be set for each number line. Once an absolute zero is established, use that reference to move the machine in a plus or minus direction along that number line.

Always think of the direction that the cutting tool (spindle) is moving. The plus or minus direction is the direction of the cutting tool, not the table. From the normal operator's position in front of the mill, when the cutting tool moves from absolute zero to the right, we will refer to this as an X+ absolute dimension, to the left of zero as X- (Fig. 4). When the tool moves from absolute zero away from the operator towards the column, it is a Y+ dimension; towards the operator is Y-. As the quill moves up from absolute zero it is Z+, down from zero is Z-. Move the machine with the handle and the numbers in the display will correspond to the tool direction.

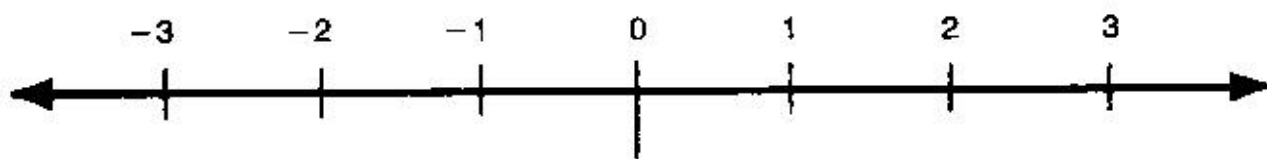
Do not proceed until all axis directions and plus and minus movements are understood. This section (Axis Direction) will be the reference for all programming motion.

B. ABSOLUTE DIMENSIONS

Figure 5 shows the X and Y axis viewed as if you were looking down at the table of the machine from above. Absolute zero can be set at any point on the table (setting zero will be covered later). At this point in time, absolute zero will be at the position where the center of the spindle was when the console was turned on.

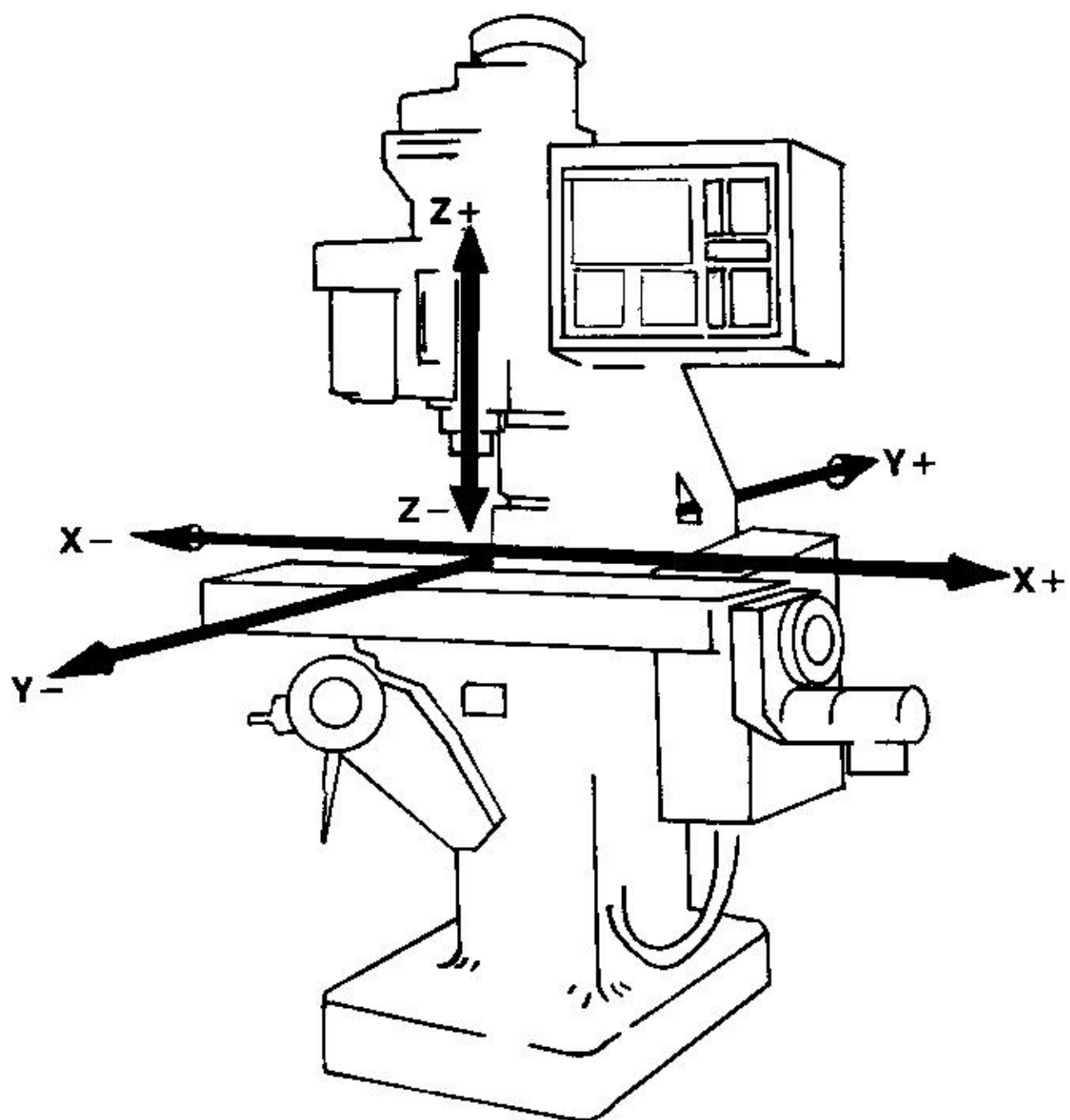
Think of the X and Y axis both starting at absolute zero and extending outwards along the axis to form quadrants. All dimensions to the right of zero are X+; to the left of zero X-. All dimensions above zero are Y+, below zero Y-.

To move the tool to a position that is to the right of zero and above zero, we must program an X+ dimension, and a Y+ dimension. The quadrants are shown with the plus and minus



NUMBER LINE

Figure 3



AXIS DIRECTIONS

Figure 4

QUADRANT DIAGRAM

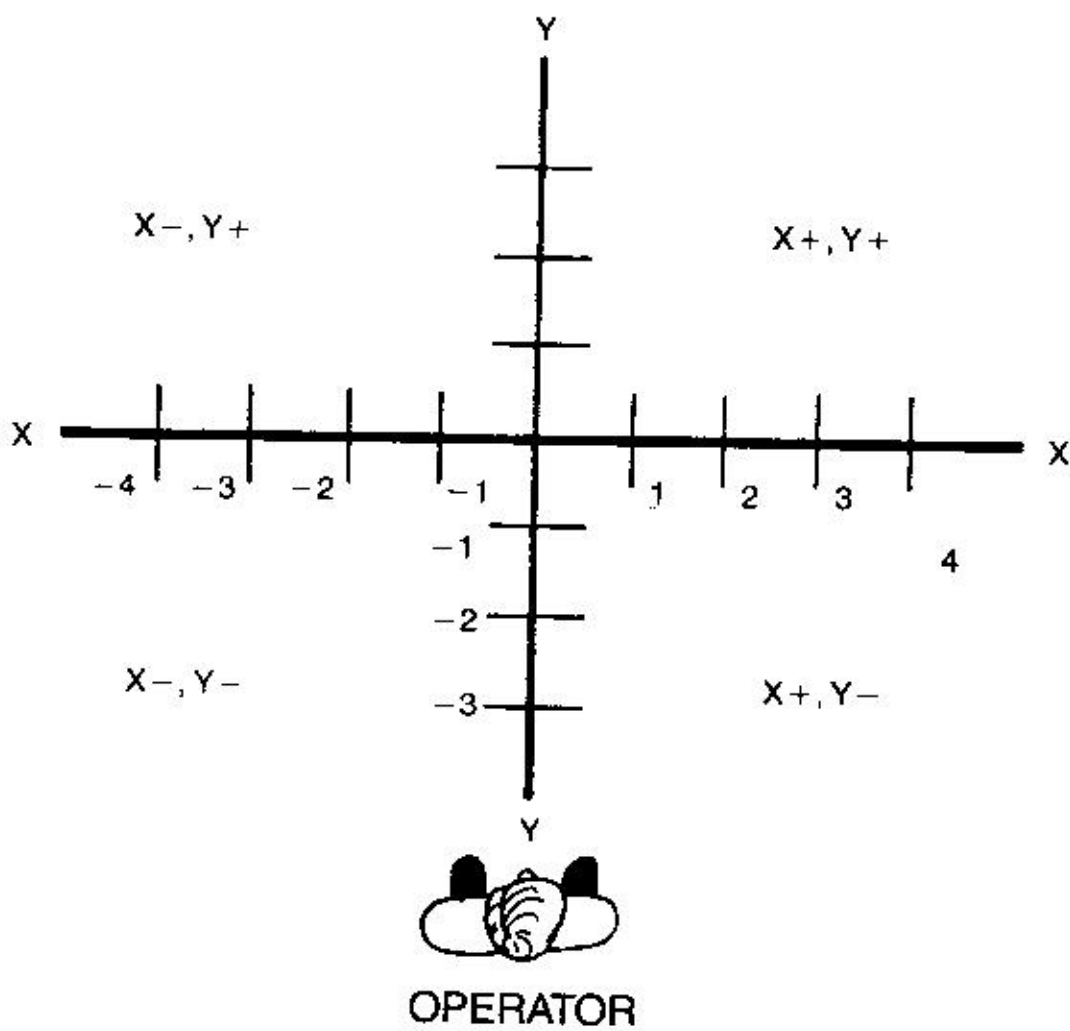


Figure 5

signs for all absolute dimensions which are in that quadrants. To move the tool to the left of zero and above it, we must program an X-dimension and a Y+ dimension.

Figure 6 shows a plate with holes dimensioned from the lower left corner. Because it is dimensioned like this, absolute zero should be set at that corner and the tool motion will be referenced to that corner. If this corner of the part was placed on the quadrant diagram, the part would be in the upper right quadrant diagram, the part would be in the upper right quadrant. This means all X and Y dimensions given to move to the hole locations will be X+ and Y+ dimensions because they are to the right and above of absolute zero. Write down the absolute dimensions for holes 1, 2, 3 and 4. Check your answers with the answers below.

Figure 7 shows a part dimensioned from a center reference. The plus and minus signs of the hole locations will be determined by their position from absolute zero just as in Figure 6. Write down the dimensions with the correct plus and minus signs for holes 1, 2, 3, 4 and 5 in figure 7. Plus signs can be omitted because they are understood by the control. If a dimension does not have a minus sign, it is assumed to be plus, and the control will show it as a plus. Check your answers below:

Absolute Dimensions For Figure 6 .

1. X.875	Y.750	ABS (Absolute)
2. X.1.750	Y.750	ABS
3. X2.625	Y1.125	ABS
4. X2.625	Y.375	ABS

Absolute Dimensions For Figure 7.

1. X-.750	Y.625	ABS
2. X .750	Y.625	ABS
3. X 0	Y 0	ABS
4. X.750	Y -.625	ABS
5. X -.750	Y -.625	ABS

C. INCREMENTAL DIMENSIONS:

Up to now, we have used the absolute system of measurement to describe tool motion. Figures 8 and 9 show a need for a different type of measuring system. Notice that there is an absolute zero and that the first hole or slot is referenced to it but the remaining holes and slots are referenced from hole to hole or slot to slot. These are known as Incremental Dimensions. They are dimensioned not from absolute zero but from the previous position.

In the top example (fig. 8), hole location 1 is located at

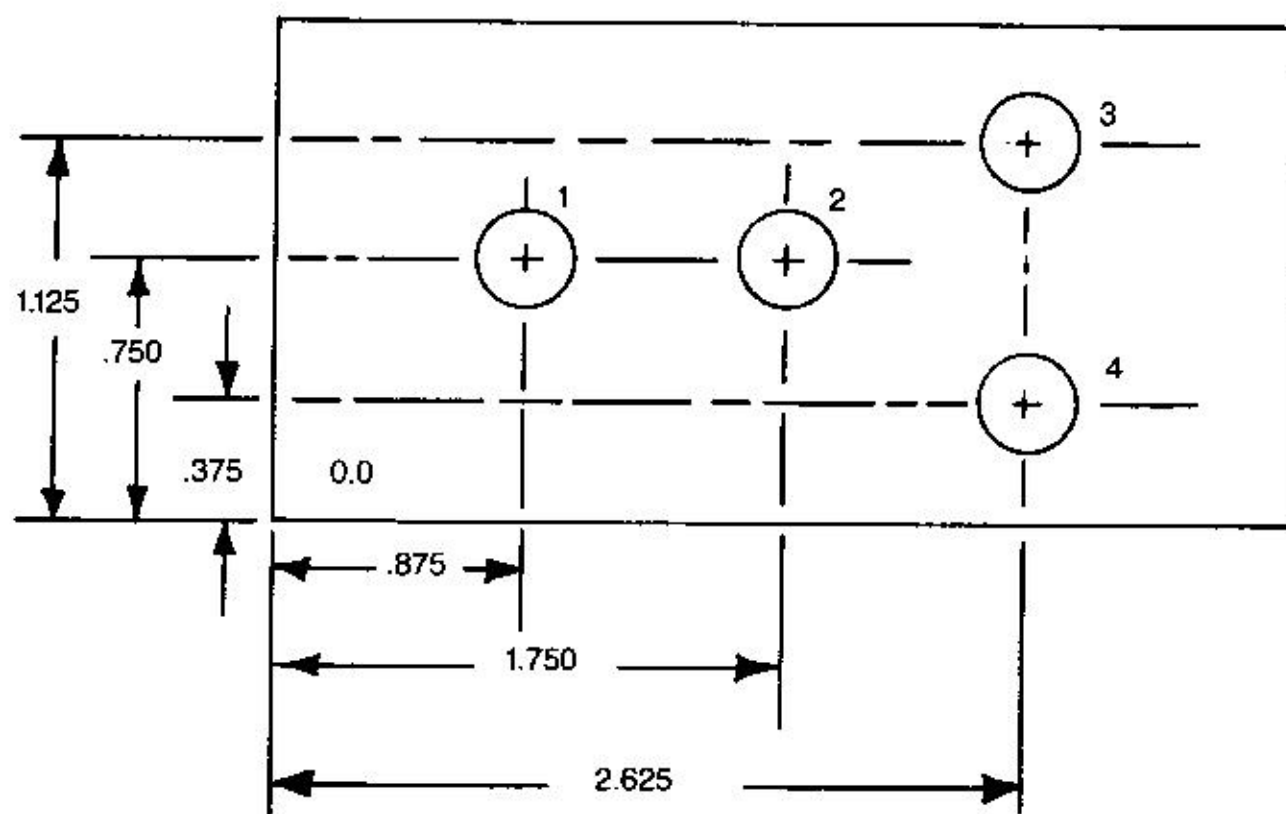


Figure 6

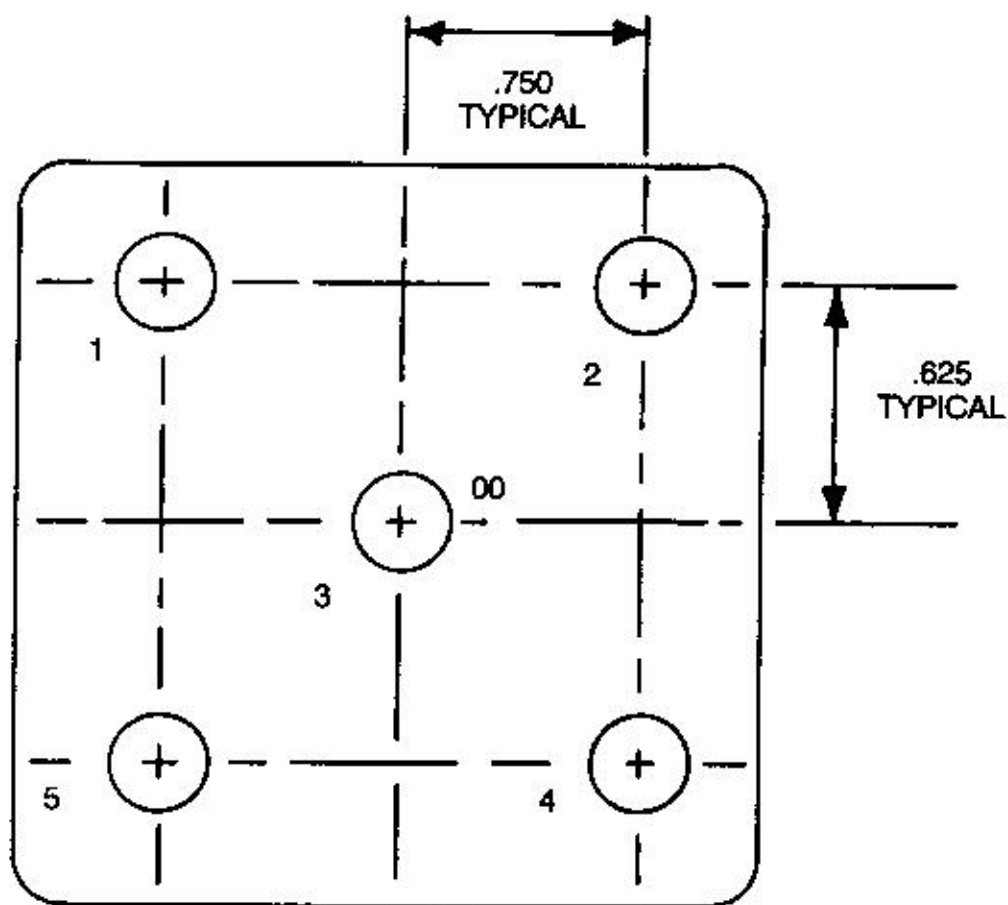


Figure 7

X+.500 Y+.500 from absolute zero. Hole location 2 is located Y+1.000 incrementally from hole location 1. Hole location 3 is located Y.500 from hole 2; hole 4 is located X3.375 from hole 3. From hole 4, we must move in the Y- (minus) direction to hole 5. The incremental distance from hole 4 to 5 is Y-.500. Similarly, hole 6 is Y-1.000 incrementally from hole 5.

In these examples, incremental dimensioning and movement do not relate to absolute zero. Incremental Dimensions are always given as a distance and tool direction. The sign of the dimension is the direction the tool is moving when using incremental.

Figure 9 shows a series of slots. Again, the right center of the top slot is given in absolute (position 1). Write down the absolute dimension of the first position and the incremental dimensions necessary to move to the remaining positions. Check your answers with the answers below.

Incremental Dimensions for Figure 9:

1. X-.625 Y-.750 (Absolute)
2. X-1.750 (Incremental)
3. Y-.312 INC
4. X 1.750 INC
5. Y-1.188 INC
6. X-1.750 INC
7. Y-.312 INC
8. X 1.750 INC

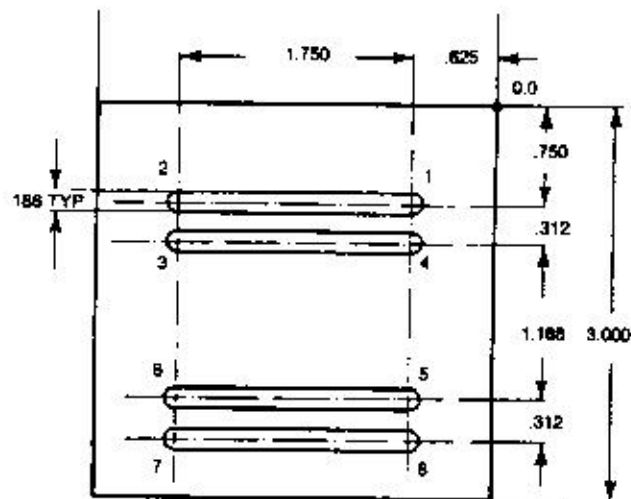


Figure 9

INCREMENTAL DIMENSIONS

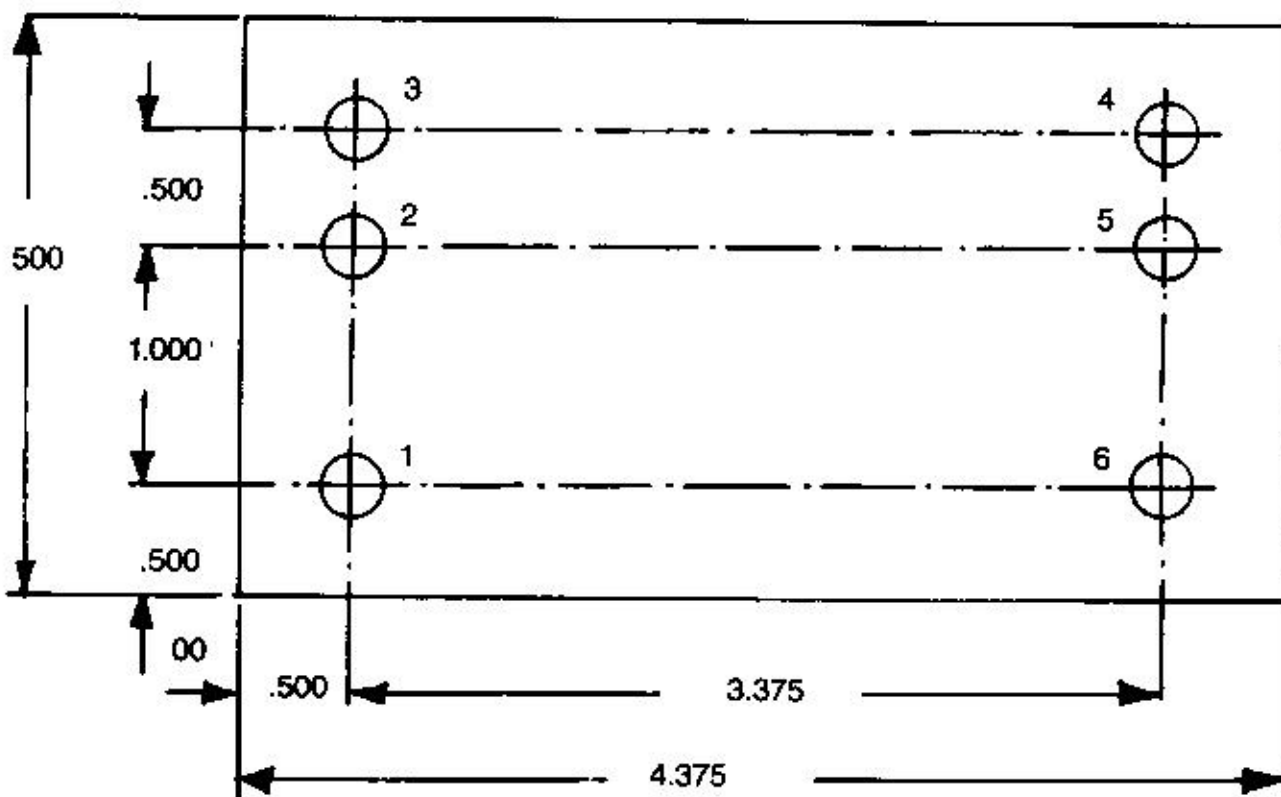


Figure 8

SECTION 3

VARIOUS MODES OF OPERATION

SECTION 3: VARIOUS MODES OF OPERATION

A. STARTING THE SERVO MOTORS

The servo will not start if one of the machine axis is at the end of travel and tripping a limit switch. Locate the switches on your machine and check to see that they are not being tripped.

The limit switches on the Z axis are behind the cover and not visible when cover is in place. These are set at the extreme top and bottom of the stroke. Move the quill with the handle or knurled knob to the center of its stroke to insure you are not tripping a Z axis limit switch.

To start the servo motors:

1. Push in then pull out **EMERGENCY STOP** button on the console.
2. Put the **MANUAL/SPINDLE/AUTO** switch, located on the back side of the servo amp cabinet, in the **AUTO** position.
3. Press the **RESET** button located on front side servo amp cabinet.

If the console is ON, the servo motors will start. No movement will take place. To turn off the motors press **EMERGENCY STOP**. This will not cause a loss of memory or position.

B. MANUAL

The Manual mode provides for manual positioning of the machine using the servo motors. Lower the knee to insure safe, unobstructed movement. If the servo motors are not on, turn them on using the instructions in sections 3A.

Now press the **MANUAL** mode button. "MANUAL" will also appear on the screen. The machine can now be moved in rapid traverse which is 100/200 inch (2540 mm) per minute (this value may be different depending on your particular machine and servo-motor combination). To move the machine, press the button of require axis to be moved, then press the **plus (+)** or **minus (-)** directional button. The machine will now move at 100 or 200 IPM for as long as you hold the button or until a limit switch is reached. Again, notice that this represents tool motion. If a limit switch is tripped, move off the trip with the hand-wheel and restart servo motors.

As the machine is moving, **HOLD IN POSN** will go out and **RUN** will appear, showing the machine is in motion. The actual speed (IPM) at which the machine is moving will be displayed on the CRT, in the middle of Area A. too center. By pressing the hand button once, the D.R.T. will change from **RAPID** to **FEED**. Feed is

40 inches (1000mm) per minute. The machine can be positioned the same way in feed using the axis buttons. Always check to make sure that you have the correct axis selected before moving. If not, press the axis button that you need to move. The active axis will always be displayed on CRT screen in Area C. Press the HAND button once again and display will read JOG 3, which is an incremental move of .100 (1.00 mm). JOG 2 is an incremental move of .010 (.1mm).

JOG 1 is an incremental move of .001 (.01mm). When any of the jogs are shown in the CRT screen, the selected axis move that increment each time the directional button is pressed.

C. SETTING THE ABSOLUTE ZERO

Zero can be set anywhere on the part. To set the X axis to zero follow the steps below:

1. Position the center of the spindle at the X0 reference point of the part.
2. Press **MANUAL** button (even if MANUAL is shown on screen) to clear any previous commands. Check bottom line on screen and ensure the correct axis is active. If both X and Y show, turn off Y by pressing the Y axis button.
3. If X is shown on bottom line of CRT, press **O ENTER**. If X is not shown as active, press **X O ENTER**.

At this time the X axis will be reset to zero and all zero's will read in the X axis display. Any axis can be set to zero using this method by pressing the correct axis button, **O** and **ENTER**. Reset each axis to zero, move the machine and notice the absolute dimension in the display will refer to this new zero.

D. PROGRAM ENTER

Program Enter is the programming mode. It is used to enter a program into memory, clear memory, add to or delete from a program, record on cassette tape or play a program into memory from cassette tape. To do any of the above you must be in PROGRAM ENTER mode. All of these functions will be discussed later in detail.

In order to get to Program Enter from Manual, first press **PROGRAM CHECK** and then **PROGRAM ENTER**.

E. PROGRAM CHECK

Program Check is used to verify what the operator has put into memory. This mode will allow you to look at 15 events at a time. The control has to be in this mode in order to use Graphics.

F. SINGLE STEP

SINGLE STEP is used to execute the program one event or move at

a time. This can be used to dry run and check a program. If the START button is pressed in SINGLE STEP mode and AUX code 1900 is active (AUX codes will be explained later in this manual) only 1 event at a time will be executed. If AUX code 1901 is active, then one X,Y move will be executed each time the START button is pressed in SINGLE STEP mode.

G. AUTOMATIC

Once the program has been stepped through and is known to be correct, the program can be run in automatic (AUTO). This is the normal operating mode for production.

H. FEED%

FEED% can be used to override a feedrate in MANUAL, AUTO, OR SINGLE STEP. If the feedrate being used is too fast, the feedrate can be reduced. This is done by pressing the button with the arrow pointing down in FEED% section. Each time the button with the arrow pointing down is pressed, the feed rate will be decreased by 5%.

Press **MANUAL**, then the **HAND** button once. Move the machine in FEED with the direction button and at the same time press the button with arrow pointing down in the FEED% section. Each time the button is pressed, the feedrate in the FEED display will decrease and the FEED% display will show the percentage of feed being used.

Similarly, if the feedrate is too slow, it can be increased. To do this, press the button with the arrow pointing up in the FEED% section. This will increase the feedrate by 5%. The feedrate can be overridden from 0-120%. The FEED override will effect all feedrates in the program.

Normally FEED% override does not effect rapid moves. There are times during set up or dry runs when it is desirable to use FEED% override for rapid moves. To accomplish this press **MANUAL AUX 1401, START**.

All rapid moves will then be overridden with FEED% override. To cancel FEED% override affecting rapid moves, press **MANUAL, AUX 1400, START**. Rapid moves will then be at 100 I.P.M. A feedrate must be in a program in order to use AUTO or SINGLE STOP.

I. INCH - MILLIMETER

The CRT will read either INCH or MM., depending upon which type of measurement is being programmed. When using canned cycles, all variables must be entered in either all inch or millimeter dimensions. Under certain conditions, it is possible to accumulate a roundoff error when using circular motion in a do loop. If this condition should arise, enter and execute the program in millimeters.

J. SOFT KEYS

These are the buttons located directly under CRT. There are a total of eight and their uses change according to which mode of operation control is in at the time. Across the bottom of CRT there are blocks which represent these buttons. These are labeled as to their uses and changes as the mode of operation is changed.

SECTION 4
PROGRAMMING_KEYBOARD: _DEFINITIONS

SECTION 4: PROGRAMMING KEYBOARD: DEFINITIONS

A. DISP POSN (DISPLAY POSITION).

When the CRT reads POSN, the screen will display the absolute position of X,Y and Z. This will be absolute no matter what mode you are in. Press **DISPLAY POSITION** to turn it off. The CRT will now read CMND and the screen will display the current programmed move in all modes except MANUAL. In MANUAL with DISPLAY POSITION off the screen will be blank because there are no programmed moves. DISPLAY POSITION can be turned ON or OFF at any time to see absolute position on the programmed move.

B. DO (DO LOOPS)

Also referred to as Step and Repeat, a do loop can be used for anything that is repeated over an equally spaced distance. Figure 11 shows a row of holes that are equally spaced at .500 along the X axis. A do loop is used to eliminate the programming of each hole. DO is entered with a number (ie., DO 5) and this is the number of times the programmed operation is repeated.

Figure 12 shows a series of windows or pockets which again are equally spaced. These represent typical examples of when a DO LOOP can be used.

C. SUBR (SUBROUTINE)

Subroutines are also referred to as mini-programs. Subroutines can be used for any section of the program which has to be repeated. In figures 13 and 14, subroutines could be used to repeat the pattern of holes or pockets. A subroutine is always stored in memory after the end of the main program and can be reached at any time for execution.

D. CALL

The Call button is used in the main program to execute a subroutine. The subroutine is assigned a number (ie SUBR 1) and then is called (ie CALL 1) to execute the subroutine. Subroutines can be labeled from 1 to 8999.

E. END

There are three uses for the End button.

1. In a do loop, End is programmed after the last event to be repeated. This stops the do loop but the program will not stop.
2. Because a subroutine is a "mini-program", it must have its own End statement. This ends the subroutine but the program does not stop.

DO LOOP

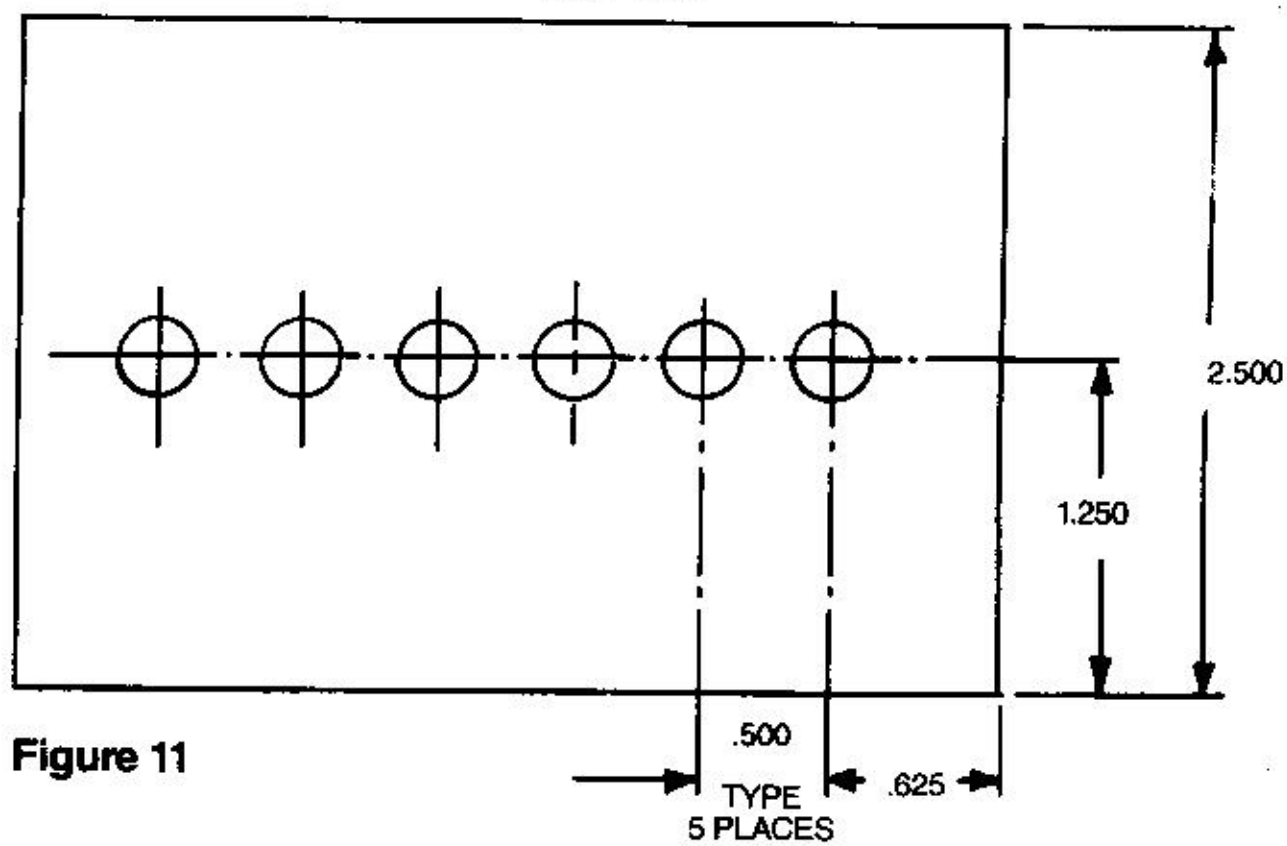


Figure 11

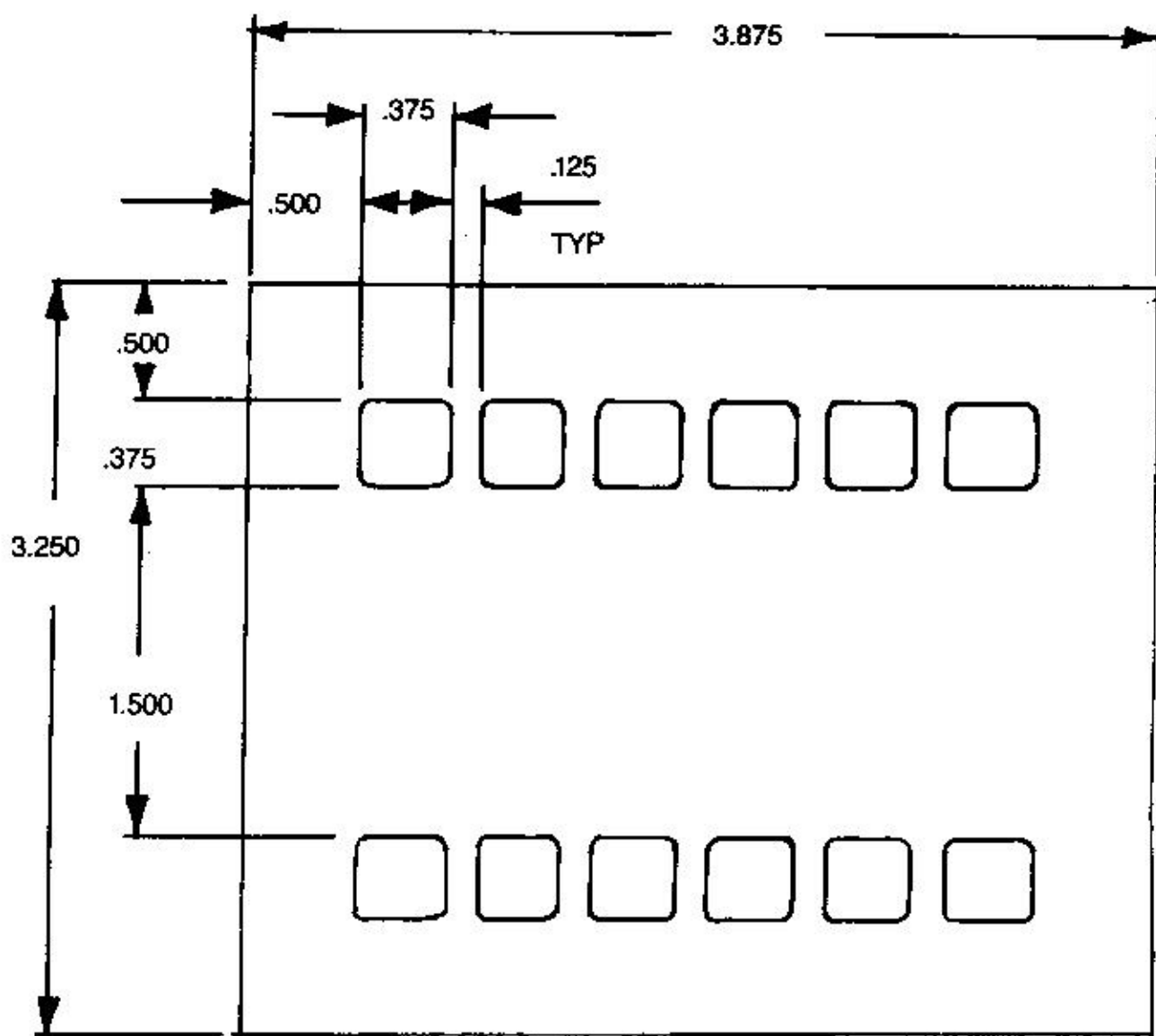


Figure 12

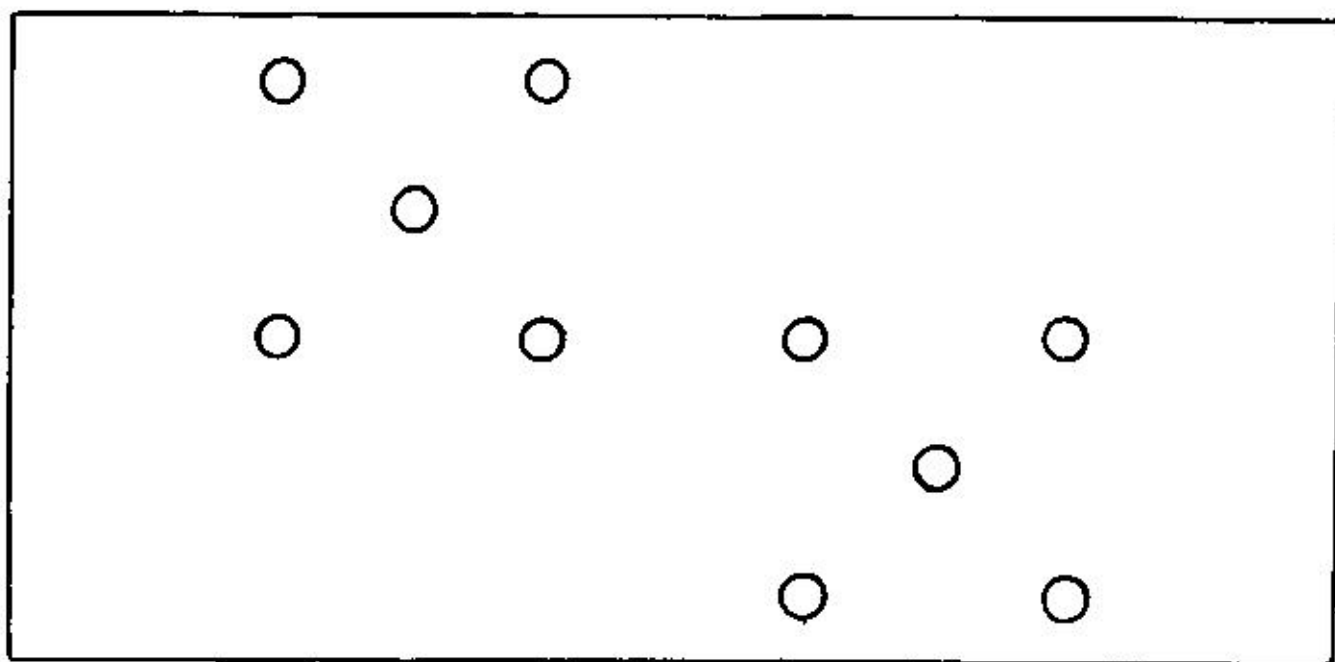


Figure 13

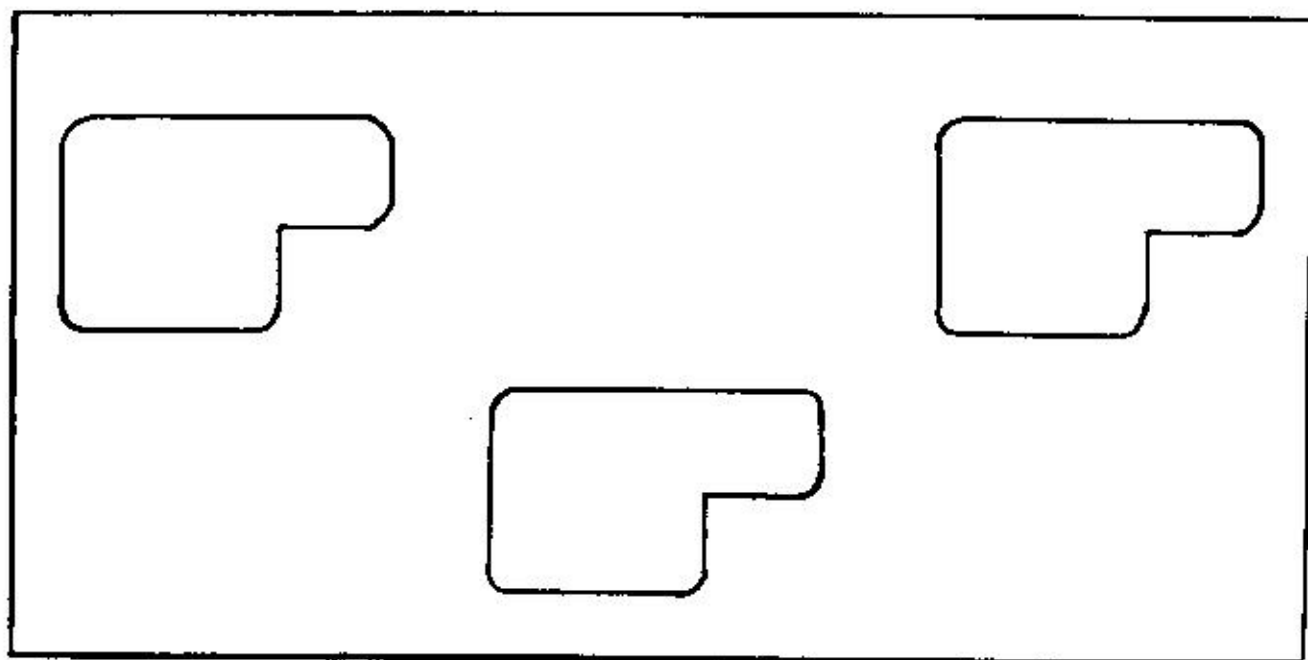


Figure 14

3. The End button is used at the end of the main program to stop motion; it also returns the program to event 1.

F. RAPID

This button is used to enter RAPID moves into the program. Rapid mode will stay active until the FEED button is pressed.

G. FEED

The Feed button has two uses. The first is to select the feedrate required and insert it into the program. The other is to tell the machine to move at a selected feedrate. At any time an X,Y or Z move is programmed, Feed or Rapid will be indicated by an F or R on the right side of the screen.

H. ABS/INC. (ABSOLUTE/INCREMENTAL)

This button is used to tell the control which type of dimension you wish to use. The second letter in a programmed line, with dimensions, will be A or I (Absolute or Incremental). Care has to be taken when programming in Incremental to ensure the correct signs (plus or minus) are used, because this determines the direction of the tool movement.

I. DWELL

The Dwell button is used as a programmed stop for moving clamps on the workpiece or turning the workpiece over. When the Start button is pressed, the program will continue.

Dwell can also be used as a timed dwell for operations such as spot facing. Timed dwells can be used for moving clamps, measuring, or changing parts.

J. AUX (AUXILIARY)

The Aux button is used in the main program to cause variations in the standard control functions. These variations are assigned numbers and entered into program prefixed by the AUX button or by using Help menu.

K. T (TOOL)

This is used in two ways: to set tool length offsets and to call for offsets in the program at the required point. This will be explained in more detail later.

L. ARC CW (CLOCKWISE)

In cutting in a circular motion, this is used to tell the machine that it must move in a clockwise direction.

M. ARC CCW (COUNTER CLOCKWISE)

Same as above, except counter clockwise instead.

N. G CODES

This button is used with canned cycles. These are programs in the control's permanent memory. The operator or programmer input information in order to make them work. These will be explained in a later section in detail.

O. V (VARIABLES)

Variables are used with the G codes and some AUX codes to input necessary information and make different canned cycles perform to your specifications.

PART ONE: CONTROL FUNCTIONS

SECTION 5

THE SERIES-M CONTROL SCREEN

PART ONE: CONTROL FUNCTIONS

SECTION 5: THE SERIES - M CRT SCREEN

When reading this section, refer to the fold-out of the Chisader Series-M faceplate (figure 1) to keep yourself familiar with the screen layout.

After reviewing the fold-out illustration, you will notice the CRT screen is divided into four general areas which are labeled A, B, C and D in the drawing. These areas will be referred to throughout this manual to help you locate specific blocks of information more quickly.

Area A tells the operator information about the active event number, actual feedrate (and percentage of programmed feedrate at which the mill is running) and, finally, the active tool number. A quick glance at Area A on your screen will tell you exactly where these pieces of information are displayed.

Area B tells the operator the X, Y and Z axis information.

Area C gives the operator information about the active mode of operation on the first line. The second line explains whether RAPID or FEED is active and explains the status (POSN or CMD) of the axis displays.

At power up, the third line of Area C reads INCH (and will change to MM for metric when that dimensioning is required). Later, when you get into absolute and incremental programming, the words ABS or INCR will appear just in front of the INCH/MM location.

The fourth line of Area C tells you that the program is in HOLD and is in position (IN POSN). At power up, there is a fifth line of information that reads COMPLETE. This means the unit was started correctly. Pressing the MANUAL button will cause this message to disappear. Later, information such as Tool Change or Error messages will appear and flash on this fifth line.

Area D shows the program. At power up, line 1 is highlighted to show this block is active. A "2" and "1" follow this highlighted number. This area will be explained in detail later in this manual.

Below Area D is a row of eight boxes with the words FORM, HOLD and MW over boxes 3, 4 and 7. These boxes and labels indicate the changing functions of the **Softkeys**. The softkey functions change according to the present mode of operation.

SECTION 6
PREPARATION FOR PROGRAMMING

SECTION 6: PREPARATION FOR PROGRAMMING

A. HOLDING THE WORK PIECE

Machining practices that were learned and have become second nature now must be put in the program. The Crusader cannot operate efficiently without the programmer and machine operator using good machine practices as the finished parts will still reflect the machinist's skill. The machinist must find the most rigid and best way of holding each work piece.

The writing of the program will depend on how the workpiece is being held. If clamps will be used, the tool must be programmed to move around them. Any obstructions will have to be kept in mind while programming to avoid collisions. How the work piece is held will determine which part dimensions are along the X-axis and which dimensions are along the Y-axis. For these reasons, the way the workpiece will be held must be decided before writing the program.

B. DETERMINING A REFERENCE POINT

The next step is to pick a point of reference (a place to set X and Y absolute zero). This will usually be the edge of the part (from which most part dimensions are given), or the center of a bore or hub on a round part. With a Crusader, the absolute zero can be set at any point. Sometimes a pin or hole in a fixture is used as an absolute zero if this is what locates the part.

Study the three examples (Figures 15-17). Figure 15 should have X and Y zero set at the top left corner. Figure 16 should have X and Y zero set at center. Figure 17 should have X and Y zero set at the large reference hole.

The way a part is held can make a difference in where the absolute zero is set. For this reason you should decide how to hold the part and where to set absolute zero before trying to program the part.

C. TOOL SELECTION

Examine the blueprint to determine what tools will be used to machine the part within the tolerances given. Next, determine the order in which the tools will be used. For example, by looking at Figure 18 we can determine that:

1. The workpiece will be held in a vise with a workpiece stop on the left side of the part.
2. Absolute zero will be set at the upper left corner because most dimensions reference that corner.
3. Tool 1 will be a spotting drill; Tool 2 a .280

DETERMINING A REFERENCE POINT

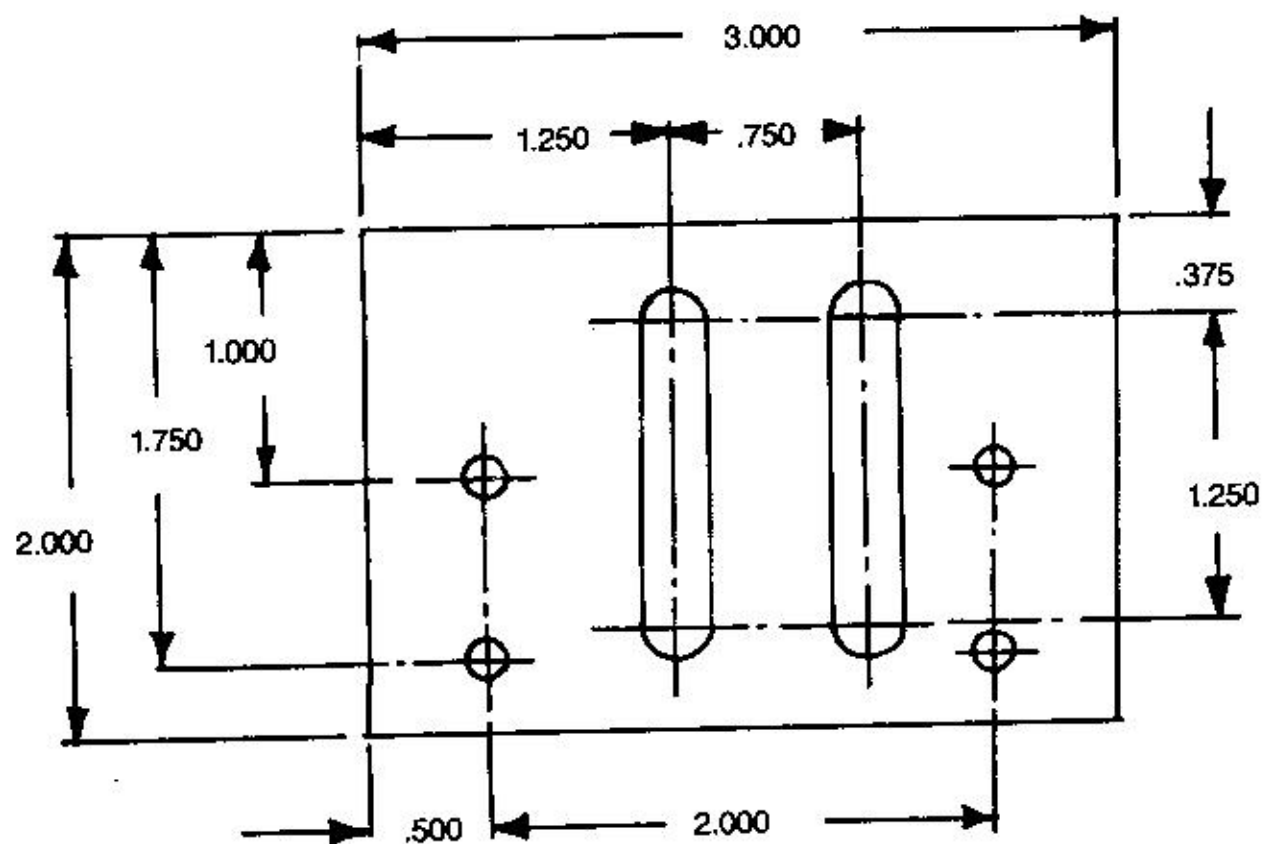


Figure 15

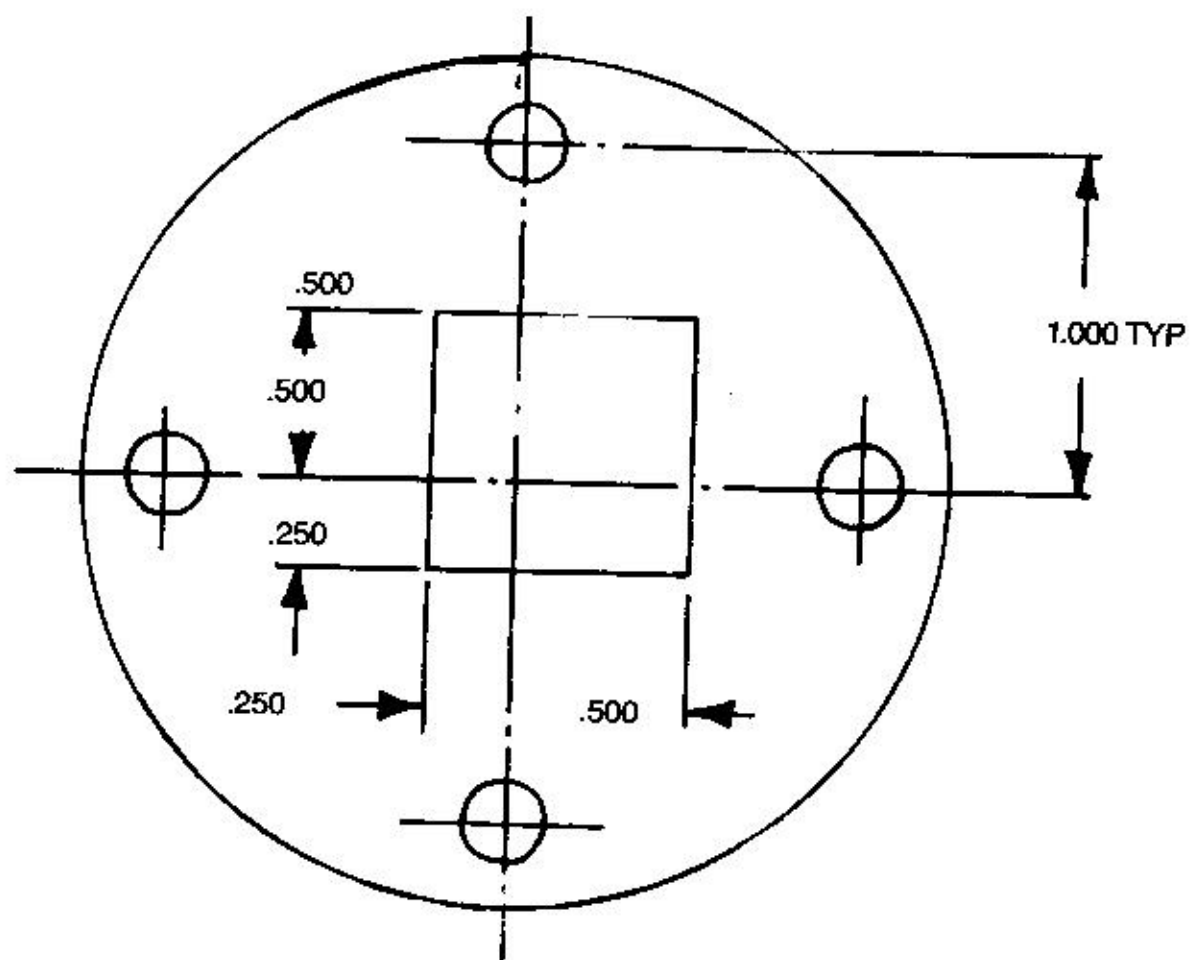


Figure 16

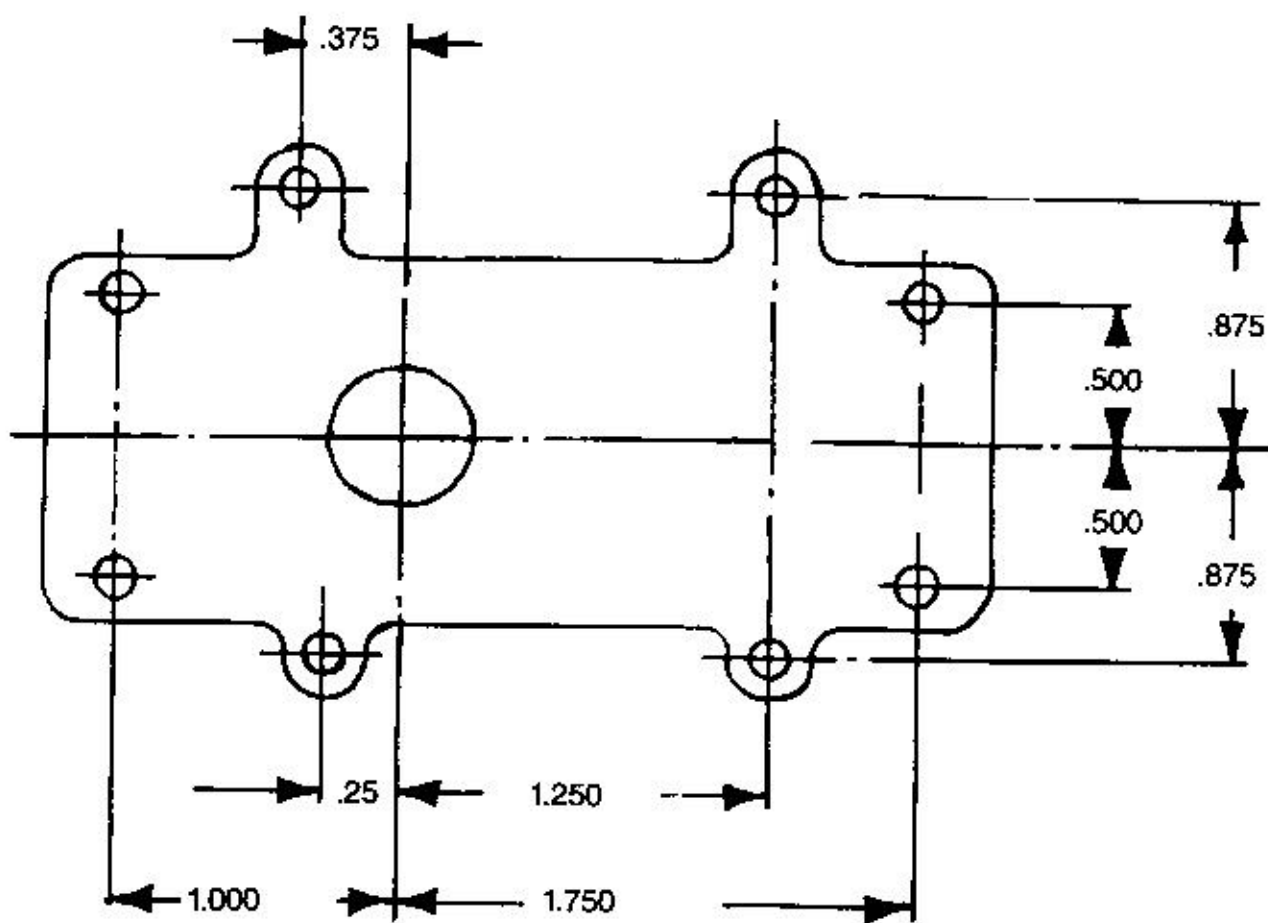


Figure 17

TOOL SELECTION

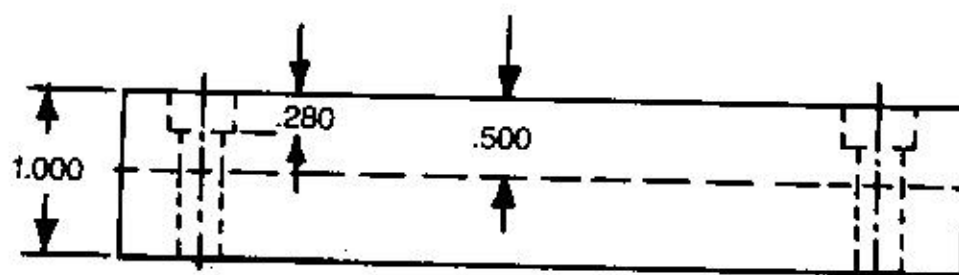
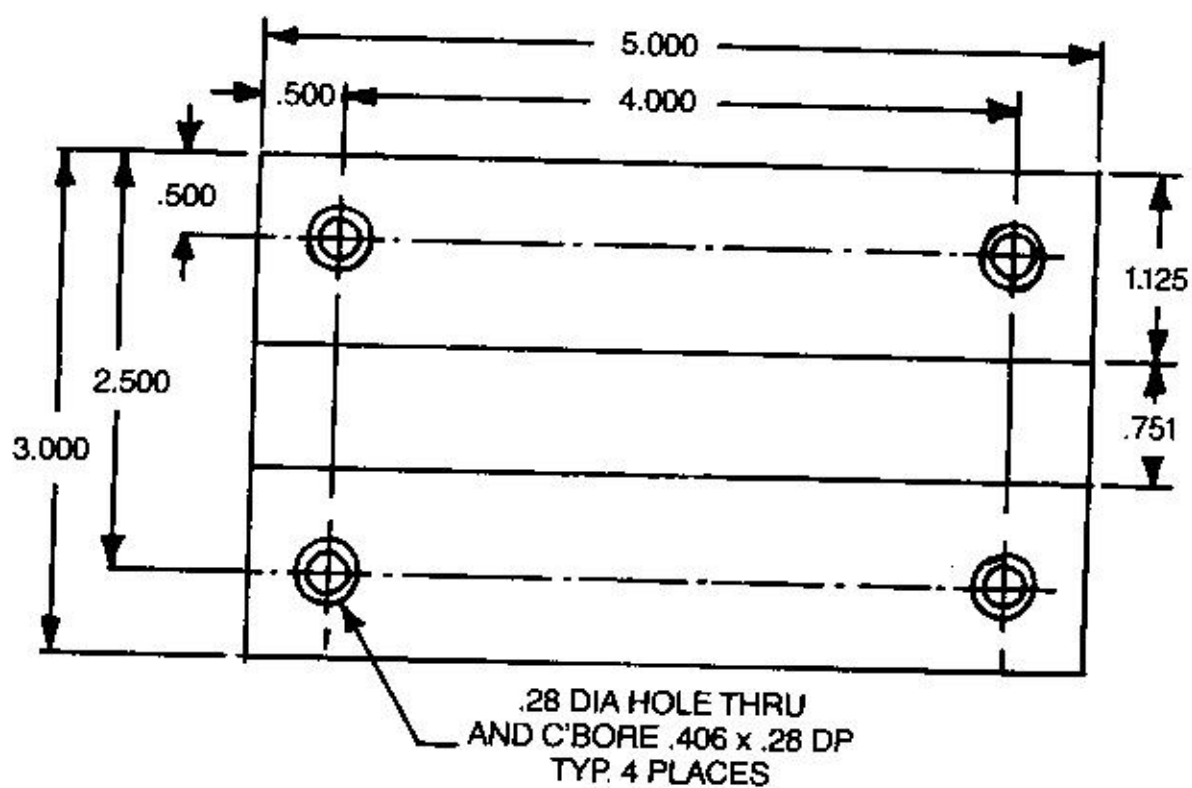


Figure 18

diameter drill; and Tool 3 a .406 end mill for counter boring (it will also be used to mill .751 slot).

The chip loads or speeds and feeds of the cutting tools are programmed in inches (or millimeters) per minute and must be within the horsepower range of the spindle. This information can be calculated by using feed and speed calculators or one of the many machining handbooks available. Tooling feeds and speeds used in the examples in this manual are theoretical but represent a practical range for the tooling used.

After the tooling is selected, a logical sequence must be determined and followed. For example, you would not drill a hole before center drilling so number the tools in the order they will be used.

D. BEGINNING A PROGRAM (ONE AND TWO AXIS LINEAR INTERPOLATION)

In this section we will write a program to move the machine from point 1 thru 5 as shown in Figure 19. You will program the X and Y axis only and use this simple exercise to become familiar with entering information. Absolute zero will be set at the lower left corner and the spindle will be programmed to move to points 1 thru 5 at 20 IPM (Inches Per Minute)

As the machine moves to points 1-5; notice you will have caused the control to execute three basic moves with the machine: An X-axis linear move, a two-axis linear move (angle) and a Y-axis linear move.

In order to keep the program in order and in understandable form, it is best to write it all down. The programming sheets provided with this manual will offer you a logical, concise, step by step format to follow. The sheets can be filed away for future reprogramming should the part ever return to your shop. Only file sheets that contain programs tested and proved in dry run.

The steps in the program are numbered starting at 1 and are called "Events". Up to 1000 fully loaded events can be stored in memory. This program will have the following events:

EVENTS	FUNCTIONS
1. Feed 20.0	This sets the feed rate at 20 IPM.
2. X0 Y0 Feed ABS	This moves the spindle to point 1 (All dimensions are absolute unless otherwise noted).
3. Y 2.000 Feed ABS	Moves spindle to point 2. Since no X axis motion is

PROGRAMMING EXAMPLE

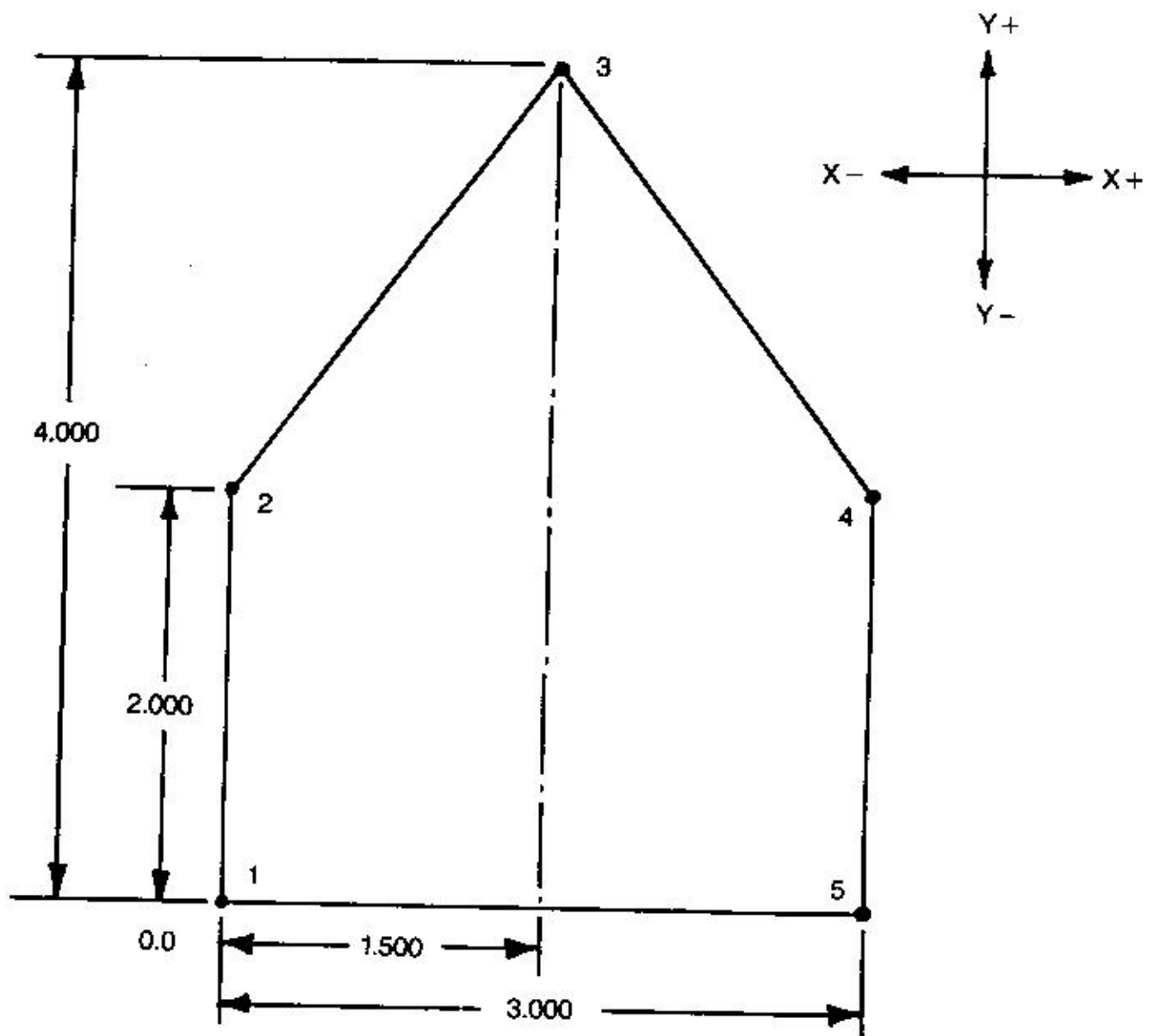


Figure 19

	programmed, the X axis does not move (single axis linear move.)
4 X 1.5000 Y 4.000 Feed ABS	Moves the spindle to point 3 Two axis linear move for angle milling)
5. X 3.000 Y2.000	Moves the spindle to point 4.
6. Y0 Feed ABS	Moves the tool to point 5. Again, no X axis motion takes place.
7. X0 Feed ABS	Moves the tool to point 1. No Y axis motion takes place (Single axis linear move).
8. END	This ends the program and returns the program to Event 1. Ready for next part to be machined.

The same moves programmed incrementally are as follows:

```

Feed 20.
X 0 Y 0      FI
Y 2.         FI
X 1.5 Y 2.   FI
X 1.5 Y-2.   FI
Y -2.        FI
X -3.        FI
END

```

Go to the control and enter this program using the instructions in the following sections.

3. CLEARING MEMORY

Before a program is entered, the previous program must first be cleared from memory. Memory can be cleared with the servo motors on or off. To clear memory:

1. Press PROGRAM CHECK.
2. Press PROGRAM ENTER.
3. Press CLEAR five times. This will clear memory. To ensure check that memory is cleared press PROGRAM CHECK.
4. Press PROGRAM CHECK.

There should be nothing showing in display. We are now ready to enter the program shown in Section D (Beginning a Program).

F. PROGRAM ENTRY

After the memory has been cleared and checked, the program can

be entered. First press **PROGRAM ENTER 1 SEARCH**. This will return the control to event 1.

NOTE

All words in bold-type represent actual key strikes.

1. To enter event 1 press: **FEED 20, ENTER**. This will store event 1 in memory and advance to event 2. Every event must be entered. As you press buttons the information will appear at the bottom of screen. When enter is pressed it will disappear from the bottom and go into the active event. If the wrong information is pressed before pressing **ENTER**, press **CLEAR**. This will clear the event so the correct information can now be put in and entered.

NOTE:

The dimensions listed in this manual make use of the decimal point key. Therefore, when "2." is seen it must be entered as "2" and "." instead of "2,0,0,0,0".

2. To enter event 2 press: **X0 Y0**. Look at the box on the right of the screen. Second line should read **FEED CMD**. The third line should read **ABS INCH**. If it does not read as shown, press **FEED, ABS** or **INCH** to bring control to correct status. Now press **ENTER**.
3. To enter event 3 press: **Y 2. ENTER**. There is no motion in X, therefore it does not require an entry.
4. To enter event 4 press: **X 1.5 Y4. ENTER**. At this time event 1 will go off screen and event 5 will show as the last line, ready to have data entered into it.
5. To enter event 5 press: **X 3. Y2. ENTER**
6. To enter event 6 press: **Y0 ENTER**
7. To enter event 7 press: **X0 ENTER**
8. To enter event 8 press: **END ENTER**

Now check the program by pressing **PROGRAM CHECK 1 SEARCH**. This will return the program to event 1. You will be able see 15 events at one time, in this program there are only 8 events therefore, the whole program can be reviewed. In the case of a longer program, to look at the next line simply press **NEXT**. If you need go back one line press **PREV** (Previous). If you need to look at the next page, press the key under the arrow pointing down on screen. To go back a page, press the key under arrow pointing up. If an error is found, see editing instructions in next section.

6. EDITING

If an error is found while in program check, press the number of the event that has the error. Then press **SEARCH, PROGRAM ENTER**. Press **CLEAR** and enter correct information.

When entering information, look at the screen before pressing **ENTER**. Check to see that everything which is suppose to be entered in the event is displayed. Also, check to see that nothing is displayed that should not be. By looking at the screen you can see exactly what is being entered, and also the three previous events. If this is done you will nearly always enter correct information.

More information on editing will be given in another section of manual. This will explain how to add and delete events.

4. PROGRAM EXECUTION

Lower the knee to insure safe, unobstructed movement. After the program has been entered and checked, use the manual jog buttons a set the absolute zero in the center of the machine travels so the axis can move far enough to execute the program.

Turn on the servo motors this has to be done before machine can be jogged. Press **SINGLE STEP 1 SEARCH**. This will return the control to event 1.

Press **START**. The control will read Event 1 which sets the feedrate. Press **START** again and the control will execute event 2. If the machine is not at absolute zero, it will feed to absolute zero.

Notice the event number that is blocked in is the active event, this will normally be the third line down. Each time the start button is pressed, one event will be executed. After the control has completed event 7 and sees that event 8 is **END**, the control will reset to event 1. Press **AUTO** then **START** to do the program in auto. If a limit switch is tripped (error message would read "SERVO DISCONNECT"), move the absolute zero to allow the machine to execute the program. The following chapters will outline several working examples. These examples will "walk you through" most of the Series M's features and will familiarize you with its capabilities.

I. CIRCULAR MOTION

Figure 20 shows an example of circular motion from point A to point B taking several paths. The path we are to program is the solid line.

Let's examine the arc we want to program, and determine what makes it unique from other arcs which go from point A to point B. The start point (A) and finish point (B) are common to each arc. If this is all that was programmed for the arc, the Crusader M would not know which path to take. The direction of the arc we want to program is clockwise.

If the start point, direction clockwise, and end point were programmed, it still would leave two possible paths. By describing the X and Y coordinate of the arc center (which also determines the radius) the last incorrect path is eliminated. It is because we must eliminate all incorrect paths that the following rules are used.

All arcs must follow this format. Each rule represents one event in the program.

1. Position the tool to the start point of the arc (X and Y).
2. Position the tool into the workpiece if it is not already at the correct depth (Z).
3. CW ARC or CCW ARC
4. Center point of ARC (X and Y)
5. Finish point of ARC (X and Y)
6. CW ARC or CCW ARC.

NOTE:

If a feedrate is not entered for the ARC, the control will use the feedrate that was last entered into the program. If no feedrate was entered previously, the machine will not run and give an error message or "ZERO FEED".

When the center point or finish point is described as an incremental dimension, the distance is given from the start point.

NOTE:

When cutting two intersecting arcs, program the finish point of the first arc as the center point of the second radius. The cutter will stop at the tangency point of the two arcs (see illustration below).

CIRCULAR MOTION

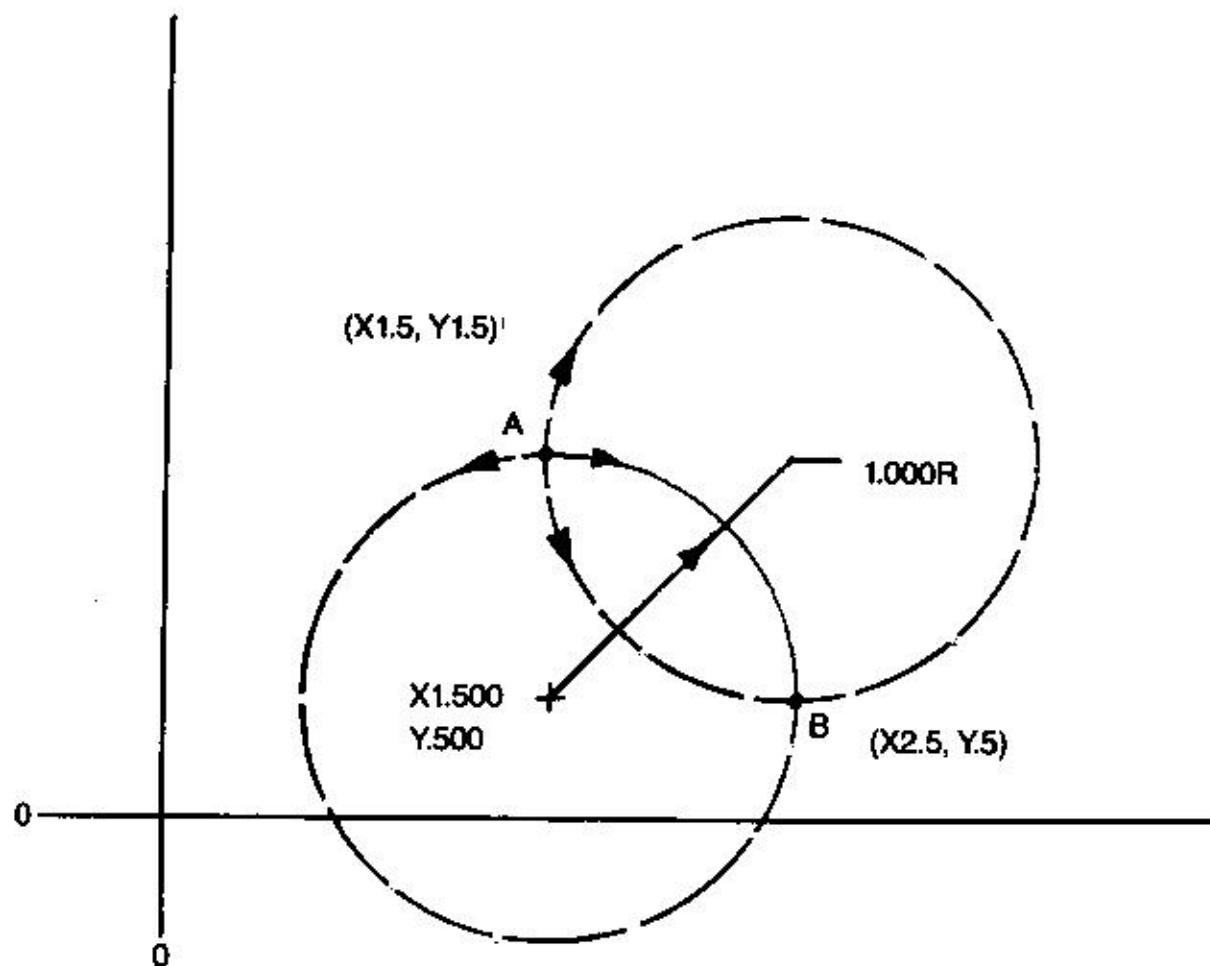


Figure 20: CIRCULAR MOTION

Use the rules stated above as guidelines and follow the program written below to see how each rule is followed.

EVENT

1. X 1.5000 Y 1.5000 R (RAPID) A (ABSOLUTE)
2. FEED 10.0
3. ARC CW
4. X1.5 Y.5 FEED A
5. X2.5 Y.5 FA
6. ARC CW
7. END

Enter and execute this example. Use figures 21 and 22 following the format given, do a full circle in absolute using a .500 end mill. Enter and execute each example below (NOTE: no Z axis move is programmed as this is a demonstration only). Remember, you are programming the center of the tool.

Full Circle (Absolute):

1. X3 Y-1.25 Position to start point.
2. ARC CW Arc Direction.
3. X 3. Y-3. Absolute position of Arc center.
4. X 3. Y-1.25
5. ARC CW
6. END

Full Circle (Incremental):

1. X 3. Y-1.25 FA Absolute position of start point.
2. ARC CW
3. X0 Y-1.75 FI The distance to the center point of the arc given from the start point of the arc.
4. X 0 Y 0 FI
5. ARC CW Arc finish command.
6. END END of program.

Half Circle (Absolute):

1. X 3. Y-1.25 FA Position of start.
2. Arc CW Arc direction.
3. X3. Y-3.00 Center Point.
4. X3. Y-4.75 Absolute position of end point.
5. ARC CW Arc finish command.
6. END End of program.

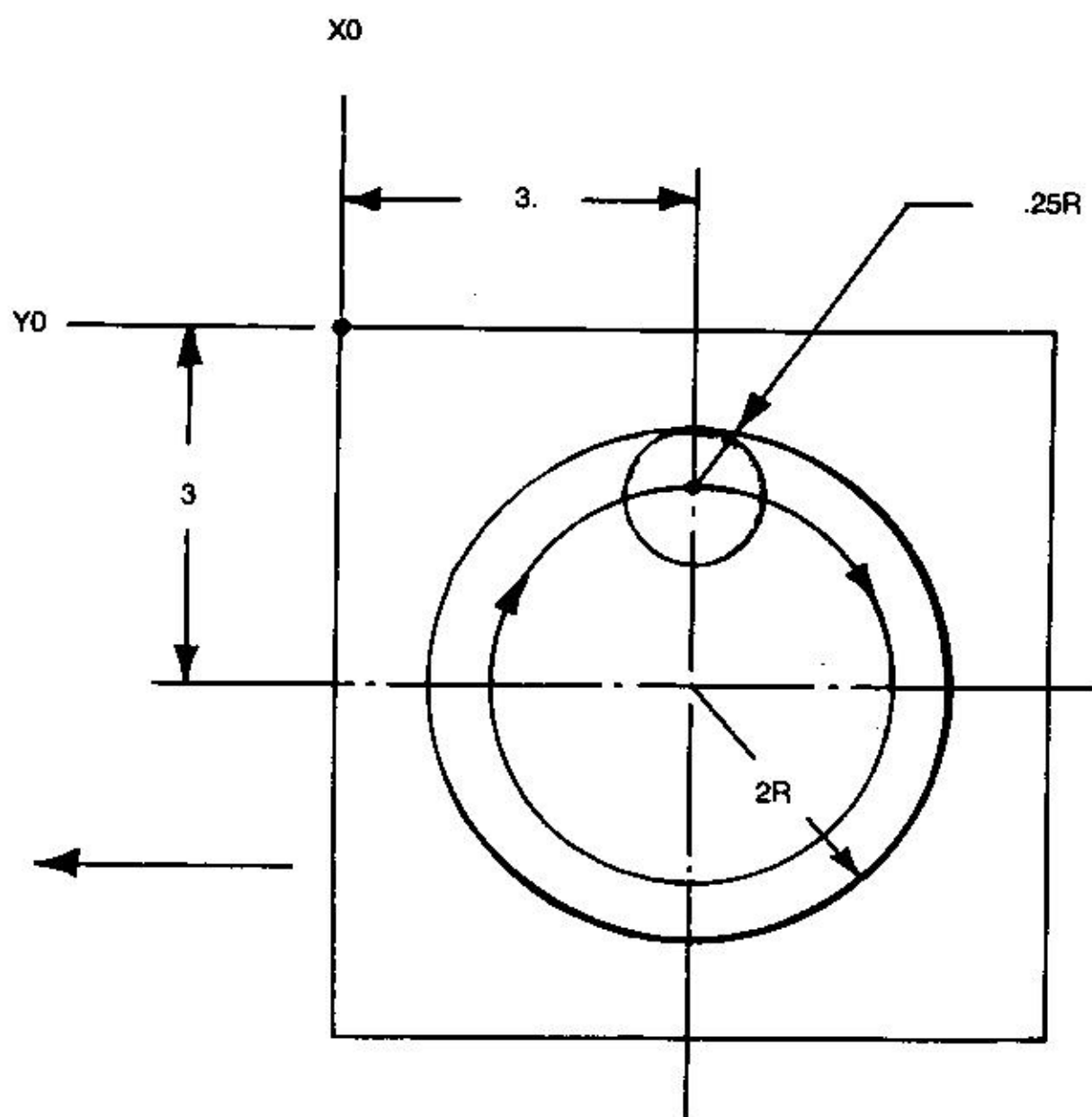


Figure 21

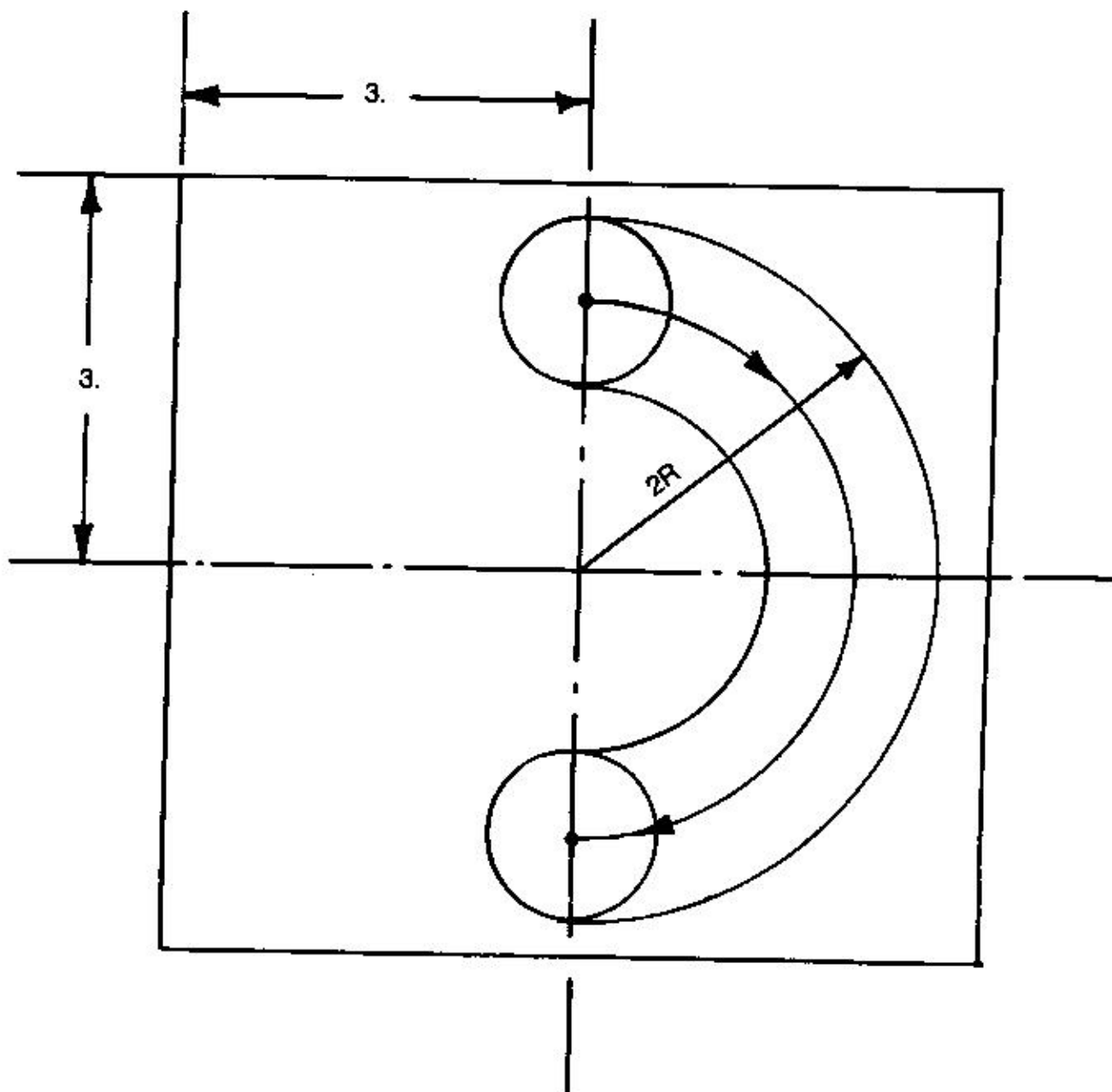


Figure 22

Half Circle (Incremental):

- | | |
|------------------|---|
| 1. X3. Y-1.25 FA | Position of start point. |
| 2. ARC CW | Arc Direction. |
| 3. X0 Y-1.75 FI | Distance to center of arc given from
start point of arc. |
| 4. X0 Y-3.5 FI | Distance to finish point given
from start point of arc. |
| 5. Arc CW | Arc finish command. |
| 6. END | End of program. |

Minimum radius that can be programmed is .002.

NOTE

If two consecutive arc's are being programmed so that the end point of the first arc is the start of the second, start point of second arc should not be programmed as a separate event.

SECTION 7
PROGRAMMING EXAMPLE
(DRILLING AND SPOT DRILLING)

SECTION 7: PROGRAMMING EXAMPLE (DRILLING & SPOT DRILLING)

A. EXPLANATION OF DRILLING AND BORING CANNED CYLES:

Drilling is programmed by using G code canned cycles. Before programming a G code certain variables must be put into the control. Also, after a G Code is programmed, a Z depth must be programmed. These canned cycles can be put into the control by two methods; either manually or by using the HELP mode. Both methods will be explained later in this chapter.

Drilling and boring canned cycles are as follows:

- G80:** Deactivates all drilling canned cycles. When the last hole is drilled must be out into program.
- G81:** Basic Drilling Cycle: Feeds to depth, Uses V20, V21 & V24.
- G82:** Counter-Boring or Spot Facing Cycle: Feeds to depth, dwells for programmed time, then rapids out. Uses V20, V21, V22 & V24.
- G83:** Peck Drilling Cycle: Feeds to peck depth, rapids out to R-plane, rapids in, feeds in, rapids out to R-plane, rapids in, feeds in, and continues repeating this process until final depth is reached. Uses V20, V21, V23 & V24.
- G85:** Boring Cycle: Feeds to depth, feeds out. Uses V20, V21 & V24.
- G86:** Boring in one Direction: Feeds to depth, program stop, rapids out, program stop. Uses V20, V21, V23 & V24.
- G87:** Chip Break Cycle: Feeds to peck depth, retracts .050", feeds to next peck, retracts .050" repeats this until final depth is reached and then rapid retracts. Uses V20, V21, V23 & V24.
- G89:** Flat Bottom Boring Cycle: Feeds to depth, dwells for a programmed time, then feeds out. Uses V20, V21, V22 & V24.

Before programming any of the drilling canned cycles specific variables have to be put into the control. These variables are listed below:

- V20:** Specifies feedrate for Z axis moves in drilling canned cycles.

V21: Specifies the heights above work piece at which tool will start to feed down. This height must be absolute.

NOTE:

When using peck cycles, always set V21 at .1000 above the actual work surface (.2mm if programming in metric).

V22: Specifies dwell time when using G82 or G89.

V23: Specifies the maximum amount of each peck in pecking cycles. This distance must be given as a positive incremental dimension. (The control will adjust the dimension to ensure that each peck is of equal distance, but will not exceed the maximum amount of each peck that has been specified by operator). Used in G83 and G87 only.

V24: Specifies the height to which the tool will rapid retract before moving to the next hole.

B. PROGRAMMING EXAMPLE:

The best way to use this example is to actually machine a blank like the one shown in figure 23 and use this program to spot drill and drill holes.

After studying example 1, it has been decided that the part will be held in a vise with a stop on the left side. The reference point on the print (X0 Y0) is shown in the upper left corner; set absolute zero's in the same place. The part will be on parallels no more than 5/16" thick.

From the blueprint it has been determined that tool #1 will be a 90 degree spot drill to center hole and tool #2 will be a .250 diameter drill.

Tool #1 will run at 3000 R.P.M. and 5 IPM.
Tool #2 will run at 2,000 R.P.M. and 4 IPM.

This information, along with the program, should always be written down before entering it into the control.

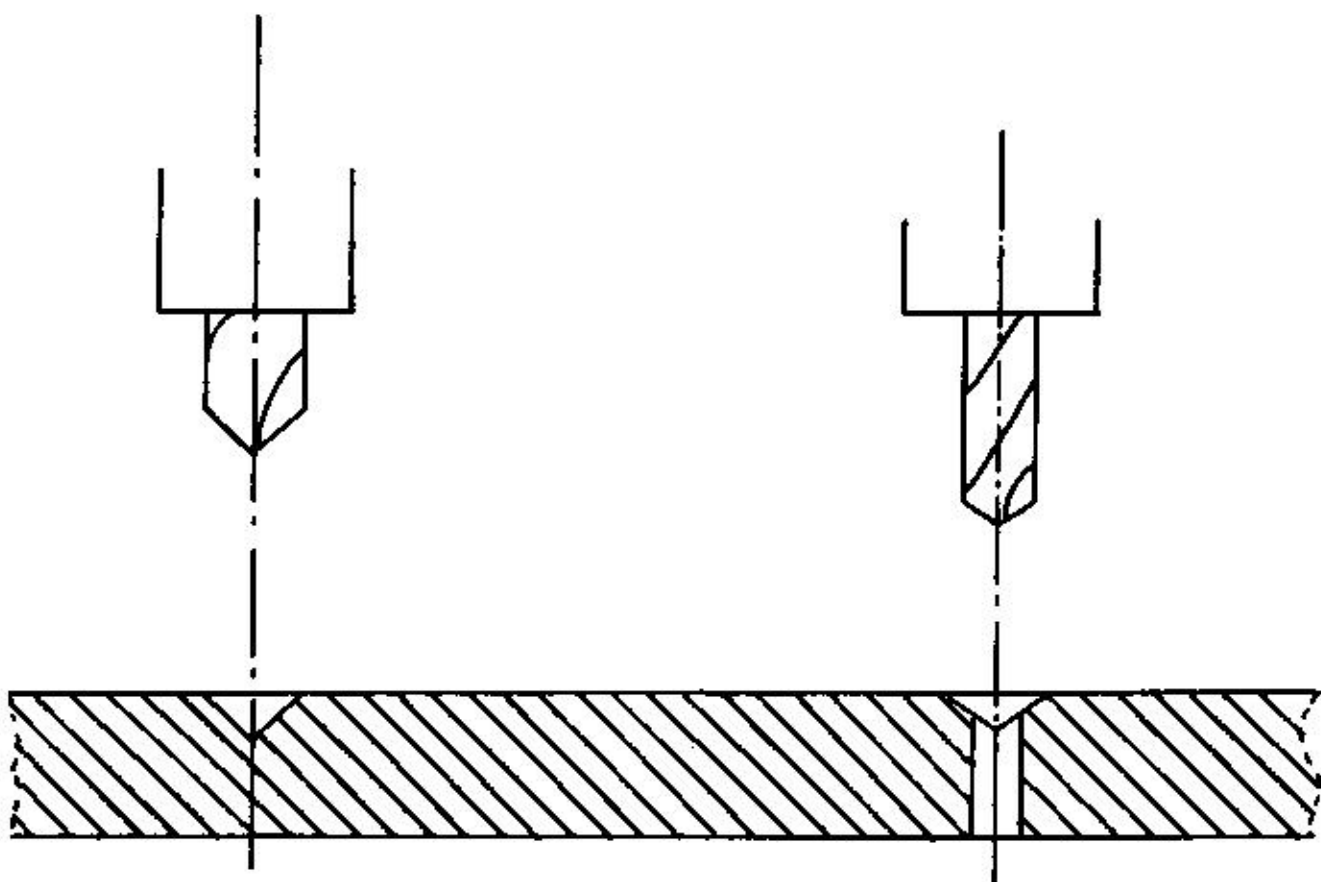


Figure 23

The machining sequence in the program will be to first list the tool and their offsets. The quill will be retracted and moved to X-3.000 Y.100. This is to move the spindle so the part can be loaded in the vise and tool #1 can be put in the spindle. The program will stop and wait for the operator to do this.

The holes will then be spot drilled. The quill will retract and move back to the tool change position. If a quick change tooling system is not available, a drill chuck can be used and the spot drill and drill can be inserted to a dead stop of the drill chuck to use as a repeatable stop.

The first section of the programming sheet has already been filled in with the setup instructions and tool list that has been established.

The steps of the program will be numbered sequentially and called "events". The event numbers are located in the first and seventh columns of the program sheet. The rest of the columns show entries to be made in each event.

It is important to establish a programming format that keeps safety in mind.

The first thing to do is create a table in the program to list the tool and their offsets. The values for tool length offsets cannot be determined by the programmer when the program is written because the length of tool is not known. Offsets are to be entered by the operator at the machine during set up.

- | | |
|-----------------|--|
| <u>EVENT__1</u> | Shows tool #1001. The "01" is the tool number. The prefix "10" is telling the control you are setting that tool offset. |
| <u>EVENT__2</u> | Is left open for entering the value of that offset by operator. X = Tool Diameter Z = Tool Length Offset. |
| <u>EVENT__3</u> | Shows tool 1002. Again, "02" represents the tool number. The prefix "10" is again telling the control that you are setting the offset for tool #2. |
| <u>EVENT__4</u> | Is left open for entering the value of the offset by operator for tool 2. X = Tool Diameter Z = Tool Length Offset |
| <u>EVENT__5</u> | Shows tool 0. This tells the control that no tool offset is active at this time. This is done at the beginning of each program to deactivate any offset. |

- EVENT__6 Shows Z0 R.A. This is done to insure tool is in Z0 retract position at top of travel. This is a safety measure to clear the part. Rapid (R) is chosen to move the quill and absolute (A) was chosen because Z0 is an absolute position. Setting Z0 will be described in more detail later in this section.
- EVENT__7 Shows X-3. Y 1. RA. This is a position off the workpiece where the operator can change tools. This should be any position that is convenient. In this case, the spindle will move to a position of X -3. (3" to left of part) and Y2. (1" to back of part) in rapid and to an absolute position.
- EVENT__8 Shows Tool #1. This activates the offset for Tool #1. This event will stop motion of machine and allow the operator to change tools. A tool change message will flash on the screen.
- EVENT__9 Shows V20 6. This is the feedrate for drilling, which is 6 IPM.
- EVENT_10 Shows V21 .1. This is the distance above top of the part where the tool will start feeding into workpiece.
- EVENT_11 Shows V24 .1. This is the height to which the tool will retract after drilling the holes.
- EVENT_12 Shows G81. This will activate the canned cycle that feeds in and rapids out to spot drill the holes.
- EVENT_13 Shows Z -.130. This is the depth to which the spot drill will feed down.
- EVENT_14 Shows X.500 Y-.500 RA. This is the position of the first hole. Because G81 is active, the tool will rapid to the X, Y location, rapid to V21 dimension in Z (.100 above workpiece), then feed at the V20 feedrate (6. IPM) to the Z depth (event 13), and finally retract to the V24 position. G81 will stay active until cancelled by G80.
- EVENT_15 Shows Y-3. R1. This is an incremental rapid move to hole 2. Since G81 is active, the spot drilling operation will be repeated and hole 2 will be spot drilled automatically.
- EVENT_16 Shows X4.5 RA. This is an absolute rapid move to hole 3. The spot drilling cycle will again be repeated.
- EVENT_17 Shows Y3. R1. This is an incremental rapid move to hole 4. The spot drilling cycle will again be repeated.

- EVENT_18 Shows G80. This deactivates the drilling cycle. This must be done after the last hole is drilled.
- EVENT_19 Shows Tool 0. This will deactivate tool 1's offset.
- EVENT_20 Shows Z 0 RA. This retract the Z-axis.
- EVENT_21 Shows X-3. Y1. RA. This will rapid the tool clear of the workpiece to tool change position.
- EVENT_22 Shows Tool 2. This will activate the offset for tool 2. The machine will stop and allow the operator to change tools. A tool change message will flash on the screen.
- EVENT_23 Shows V20 4. This is the feedrate for drilling.
- EVENT_24 Shows V21 .1. This is the distance above the workpiece at which the tool will start feeding into the part.
- EVENT_25 Shows V24 .5. This is the height to which the tool will retract before moving to the next hole location. At the next hole location the tool will rapid down to V21 height.
- EVENT_26 Shows G81. This activates the canned cycle to feed in, rapid out, to finish drilling holes.
- EVENT_27 Shows Z-.35 FA. This is the depth to which the drill will go on the Z axis.
- EVENT_28 Shows X.500 Y-.500. This will rapid the tool to the XY location of the first hole. Because G81 is active, the tool will rapid to .5 (V24) above the part. It will then rapid to .1 (V21) above the part, feed at 4. IPM (V20) to Z-.35 and rapid back to .5 above the part.
- EVENT_29 Shows Y-3. RI. This is an incremental rapid move to hole 2. Because G81 is active, the second hole will be drilled as described.
- EVENT_30 Shows X4.5 RA. This is an absolute rapid move to hole 3. Again the drilling cycle will be automatically repeated.
- EVENT_31 Shows Y 3. RI. This is an incremental rapid move to hole 4. Again the drilling will be repeated.
- EVENT_32 Shows G80. This deactivates the drilling cycle to stop the feed in, rapid out motion.
- EVENT_33 Show Tool 0. This will deactivate tool 2's offset.
- EVENT_34 Shows Z0 RA. This will retract the quill.

ANILAM CRUSADER PROGRAM SHEET

ART OR DRWG. # EXAMPLE I PROGRAMMER J T M DATE 8-1-85

TOOL LIST

OFFSETS

Tool 1: SPOT DRILL @ 3,000 RPM
 Tool 2: .250 DRILL @ 2,000 RPM

SETUP INSTRUCTIONS

WORKPIECE HELD IN VICE WITH X, Y AND
 "O" IN UPPER LEFT CORNER OF WORKPIECE.
 STOP IS ON THE LEFT SIDE OF PART.
 TOOL OFFSETS ARE SET AT THE TOP
 OF THE PART.

VENT NO.				RAPID /FEED	INCR /ABS	EVENT NO.				RAPID /FEED	INCR /ABS
1	TOOL	1001				21	X-3.	Y 1.		R	A
2	Y O		Z TLO			22	TOOL	2			
3	TOOL	1002				23	V 20	4.			
4	X.25		Z TLO			24	V 21	.1			
5	TOOL	φ				25	V 24	.5			
6			Z φ	R	A	26	G81				
7	X-3.	Y 1.		R	A	27		Z-.35		R	A
8	TOOL	1				28	X.5	Y-.5		R	A
9	V 20	6.				29		Y-3.		R	I
10	V 21	.1				30	X4.5			R	A
11	V 24	.1				31		Y 3.		R	I
12	G81					32	G80				
13			Z-.13	R	A	33	TOOL	φ			
14	X.5	Y-.5		R	A	34			Z φ	R	A
15		Y-3.		R	I	35	X-3.	Y 1.		R	A
16	X4.5			R	A	36	END				
17		Y 3.		R	I	37					
18	G80					38					
19	TOOL	φ				39					
20			Z φ	R	A	40					



- EVENT_35** Shows X -3. Y1 RA. This rapids the tool to the tool change position.
- EVENT_36** Shows END. This finishes the program and resets the Series-M to event 1. The machine will stop when this event is reached.

C. ENTERING PROGRAM INTO CONTROL

Now that the program is written, it is time to put it into the Series-M control. Just follow the step-by-step instructions below.

When the control is started up it will come up in manual. Press **PROGRAM CHECK** and then **PROGRAM ENTER**. The screen will read **PROGRAM ENTER** on line 1, Area C. Next, press the **CLEAR** button 5 times. This will clear memory and return control to Event 1.

The program is to be entered exactly as on the program sheets so use them to follow this sequence. Since this program is dimensioned in inches be sure line 3, Area C reads inches.

- EVENT__1** Press T 1001 and **ENTER**. This stores the display information into memory and advances to next. Notice that there are two event 1's. When T1001 is programmed it appears on the lower of the two lines. When **ENTER** is pressed the information will transfer to the top line and **EVENT 2** will be ready to receive data.

NOTE:

If wrong information is pressed before pressing **ENTER**, just press **CLEAR** once. This will clear the event so the correct information can now be entered.

- EVENT__2** A tool diameter must be entered in the X-axis. Press X.3125 and press **ENTER**. Later, a Z dimension will be entered in this event for the tool length offset of tool #1.
- EVENT__3** Press T1002 **ENTER**.
- EVENT__4** Press X.25 for tool diameter and **ENTER**. The tool length offset for tool #2 will be entered later.
- EVENT__5** Press T 0 **ENTER**
- EVENT__6** Press Z 0. Line 2, Area C should read **RAPID**. If it does not, press **RAPID**. The third line should show **ABS (Absolute)**. If not, press the **ABS/INC** button. Press **ENTER** when all information is correct. This line of information should look as follows Z +.000 R A.

EVENT__7 Press X -3. Y1. Notice Area C still shows Rapid and Absolute. Press ENTER.

EVENT__8 Press T 1. ENTER.

NOTE:

The next 5 lines will be put in using HELP MODE. Press HELP; press B; press B1. The control will ask for the feedrate (V20). Press .1 ENTER. The R plane (V21) is .1 and the Z plane (V24) is .1. Control will now put in G81 and ask for Z depth (which is event 13 in the program). Press -.13 ENTER.

Now all questions are answered. Press ENTER and control will revert back to the normal programming screen at Event 14.

EVENT_14 Press X .5 Y-.5 ENTER.

EVENT_15 Press Y -3. Press ABS/INC button. The letters on the end of this line should read RI. If so, press ENTER.

EVENT_16 Press X 4.5 and then press the ABS/INC button. The letters on the end of this line should read RA. If so, press ENTER.

EVENT_17 Press Y3. and then press ABS/INC. The end of this line should read RI. If so, press ENTER.

EVENT_18 Press G80 ENTER.

EVENT_19 Press T 0 ENTER.

EVENT_20 Press Z 0 and then press ABS/INC. The end of this line should read RA. If so, press ENTER.

EVENT_21 Press X-3. Y1 ENTER.

EVENT_22 Press T 2 ENTER.

EVENT_23 Event 23 through Event 27 will be put in using HELP MODE. Press HELP G81. Then fill in the following variables:

 V20: 4. ENTER

 V21: .1 ENTER

 V24: .5 ENTER

G81 is automatically entered. The Z depth is -.35 ENTER.

EVENT_28 Press X.5 Y-.5 ENTER.

EVENT_29 Press Y -3. INC ENTER.

EVENT_30 Press X 4.5 ABS ENTER.

EVENT_31 Press Y 3. INC ENTER.
EVENT_32 Press G80 ENTER.
EVENT_33 Press T0 ENTER.
EVENT_34 Press Z0 ABS ENTER.
EVENT_35 Press X-3. Y1. Press ENTER.
EVENT_36 Press END ENTER.

D. CHECKING THE PROGRAM

After a program has been entered into the control, it should always be checked before attempting to run the part. To check the program, press **PROGRAM CHECK 1 SEARCH**. You can now see the lines 1 to 15. If the information is correct in these lines press the button under the arrow pointing down. The control will now show lines 15 to 29. Compare each event with your program sheet to ensure the information is correct.

By pressing the button under the arrow pointing up, the program will scroll back a page (ie, if you're on line 15 to 29, press arrow pointing up and you will get lines 1 to 15). If you only want to move one event at a time press **NEXT** to go forward, **PREV** to go back one event.

If a mistake is found while in Program Check, then press **PROGRAM ENTER**, the appropriate event number, and **SEARCH**. The event in error will appear on the bottom of the screen and can now be corrected. Pressing **ENTER** returns the screen to Program Check mode.

To edit an axis dimension, the control must be in Program Enter mode and the line that needs editing must be at the bottom of the screen. Press the axis that needs to be changed twice (the first press clears the old dimension, the second press reactivates the axis). Now, insert the new dimension and press **ENTER**.

In case a Tool, G Code, Aux code, V code, Dwell, Call or Subroutine needs to be changed, press **CLEAR** and input the new information. Now, press **ENTER**.

E. ADD EVENT

To add a single event, put the control into Program Enter mode. Select the Event at which you need to add an event. Press **INSERT ENTER**. This will open a gap in the program and shift every event after this down by one (ie. if inserting at EVENT 12, this will now become event 13 and 13 will be 14 and so on).

In case more than one event needs to be inserted (ie., 10 events need to be inserted at Event 9). Press **INSERT 9 INSERT 10 ENTER**. This will open up a 10 event gap in the program and will move all events behind this up by 10.

If the Help menu is used to add events, then you do not need to open a gap. Just bring the event (that you need to insert from) to the bottom of the screen. Now, press **HELP** and select the G code or AUX code required and enter information as previously described. When all information has been inserted and **ENTER** is pressed, the Series - M will automatically insert this data into the program.

F. DELETE EVENT

To delete one event set the control to Program Enter mode. Bring the event to be deleted to the bottom of the screen, press **DELETE ENTER**, this event will now be removed from program. All events behind this will move up by one number.

The control can also Mass Delete. For example, to delete lines 10 through 20, use the following procedure: Go to Program Enter mode. Press **DELETE, 10, DELETE, 20, ENTER**. This will delete lines 10 through 20.

G. MACHINE SET UP

The head of the machine should be trammed 90 degrees to the X and Y axis and the vise should be parallel to the X axis. A workpiece stop should be on the left side.

Place the part in the vise and against the workpiece stop on parallels not thicker than 5/16. Put an edge finder in the spindle. Using the hand wheels or the manual jog buttons, find the left edge of part. Press **MANUAL**, then set zero by pressing **X, 0, ENTER**.

Raise the edge finder above the top of the workpiece and, using the readout display, move the edge finder a distance half of its diameter over the workpiece. Set Y0 by pressing **Y, 0, ENTER**. If you move over the too left corner of the part, the readout will display X 0 and Y 0.

H. SETTING TOOL LENGTH OFFSETS

Tool offsets can be set with motors on. If the motors are, the quill will have to be moved manually.

To set the tool length offsets for programming example 1, follow the step-by-step instructions below:

1. Put the spot drill and .250 drill each in their own quick change holder.
2. Turn on the servos and press **MANUAL**, even if the control already reads Manual, to clear any previous commands.
3. Jog the quill up near the top limit. If the limit switch is tripped, lower the quill by hand until you are just off the top limit switch and re-start the servos.
4. Press **TOOL 0 START**, to ensure no offset is active.
5. Press **Z0 ENTER** to set this as the Z0 full retract position.
6. Place the longest tool in the spindle and set the table height using the knee. Leave about 1/2 inch between. The tip of the tool and the top of workpiece.
7. If the tool in the spindle is not tool #1, replace it with tool #1.
8. Jog the tip of the tool down to the top of the part. To do this select Manual mode press Z, and select the type of movement required (rapid, feed, jog 3, jog 2 or jog 1). Jog 3 would be good to start, so press the **Hand** button until the screen reads JOG 3 on line 2, Area C. As the tool gets closer to the part, jog increments should be changed.
9. Press **PROGRAM ENTER**. Line 1 in Area C should read **MANUAL ENTER**. Press 2 **SEARCH**, this will display event 2 (which is empty) on the bottom of the program.
10. Press **DISP POSN**. This will turn on the display and show the absolute position. Press **Z ENTER**. Check the bottom line of the program ensure that only the Z axis is lit. If X and/or Y is lit, then press the X and/or Y button. This will take the absolute position in the Z axis display and enter it as tool #1's offset in event #2. This is our distance of compensation for tool #1. The control now knows where the top of the part is when using tool #1.

11. Using the jog buttons, raise the tool off the workpiece and place tool #2 in the spindle.
12. Jog tool #2 down to the top of the part.
13. Press **4 SEARCH**. Press **Z ENTER**. This enters the absolute position of Z axis as the offset for tool #2 in event 4.

I. REVIEW OF PROGRAMMING FORMAT

Let's review our procedure so far, as this should be a standard format for each job to be done with any CNC.

We first examined the print to find the best way to hold the part. We then selected a point to set absolute zero, selected our tooling, and thought about a logical sequence. Next, we wrote the program, using a format of retracting the drill and moving to a tool change position first.

Finally, we entered the program into the memory and checked it to see if it was entered correctly. X and Y zero was set, then the tool length offsets were entered. If these steps have been followed correctly, we are ready to run the part in a dry run graphics mode.

K. CHECKING PROGRAM WITH GRAPHICS

In order to run graphics, the **EMERGENCY STOP** button must be pulled out. There has to be a program in memory, tool 0 must be the active tool, and the control has to be in program Check mode. If tool 0 is not active, press **MANUAL T 0 START**.

To enter graphics, press the soft key which has the word **DRAW** above it (far left soft key). We are now in graphics mode and have five parameters to set.

1. The first parameter that needs to be set is the position of absolute zero. Press **1** and the screen will give several options for zero placement. For this part we need position #2 (upper left corner), so press **2** and a "cross" will appear in top left corner of the screen. This shows you where zero is set. Now, press **ENTER**.
2. In order to fit part into the screen area a scale size needs to be determined. Press **2**. At startup the scale size will be set to 1, but for this part we need to set the scale to .8, so press **DELETE .8**. Check to see that you have entered correct value and press **ENTER**.
3. The next parameter to set is Start Event and Finish Event. So press **3**. At start up the starting event will be set at 1, which we need. However, the finish event will be set at 999 which is incorrect. Press **NEXT** and the cursor will move down to Finish Event. Press **DELETE 36 ENTER**. Check to see that you have entered this number correctly and press **ENTER**.

These are the only parameters that need setting for this part. We can now run graphics, but first ensure that the **Emergency Stop** is out.

Press **START** and the control will now start to run graphics. If servo motors are running they will automatically turn off at this time.

If **HOLD** button is pressed the graphics will stop. They can be restarted by pressing **START**. By pressing **SINGLE STEP** the graphics will run one step at a time. When a **STEP** message is flashed in bottom right corner, press **START** to go to the next move.

Views may be changed at anytime by pressing **AUX 1** (X,Y view), **2** (X Z view) and **3** (Y Z view).

When graphics are complete press **END** to get back to Program Check.

Light lines are rapid moves. Dark lines are feed moves. When drilling, and in XY view, holes will show as a cross on the screen.

K. RUNNING THE PROGRAM

Now that we have checked the program using graphics and it looks correct, we can run it on the machine. Remove the part from the vise. Be ready to press **EMERGENCY STOP** while doing this dry run.

Turn on the servo motors. Press the **SINGLE STEP** button. Then press **1 SEARCH**. This will insure that we start from the beginning of the program. Press the **START** button to execute events 1-6. Each time the start button is pressed the control will execute one move of the program.

When the sixth event is executed, the quill will retract if not already retracted. Event 7 will move the quill to the tool change position. Event 8 will display that the operator is to put tool 1 in the spindle (the program sheet tells us that the tool should run at 3,000 RPM).

Turn on the spindle and press **START** to proceed through each step of the program. Check each move to see that the machine moves to the correct position at the correct feedrate. When tool 2 is called in the program, put it in the spindle and continue through the program. If errors are found, edit the program.

Once this is done, put the machine in **AUTO** and put the part in vise. Press **1 SEARCH START**. The machine will move to the tool change position and stop. Put Tool 1 in the spindle, turn it on and press **START** to center drill part. After Tool 1 is

finished the quill will retract and the machine will go to the tool change position. Change the tool.

Press **START** and the part will be drilled. On completion of drilling, the quill retracts and machine will go to tool change position and reset program to Event 1.

SECTION 8
MORE DRILLING EXAMPLES

SECTION 8: DRILLING EXAMPLES

This section will explain in detail how the operator may use the crusader Series-M control to drill into a workpiece. Several features and functions are explained; such as peck drilling, co loops, subroutines, etc.

A. DRILLING FORMAT RULES

When programming the G80 drilling cycles, these are six basic rules that must be followed. The example in figure 24 will be used to illustrate these rules in practical form.

<u>EVENT:</u>	<u>FUNCTION:</u>
1. Tool 1001	Offset for Tool 1.
2.	Offset value entered at set up.
3. Tool 0	De-activate offsets.
4. Z 0 RA	Retract quill.
5. X-1. Y1. RA	Rapid to a tool change position.
6. Tool 1	Activate Tool 1 offset, program stops.
7. V20 8.	Set drilling feedrate.
8. V21 .100	Set drilling start height.
9. G81	Activate drilling.
10. X1. Y1 Z-.50 RA	First hole location and Z depth.
11. X2. Y2.	Second hole location.
12. X3. Y3.	Third hole location
13. G80	De-activate drilling.
14. Tool 0	De-activate tool offsets.
15. Z0 RA	Retract quill.
16. X-1. Y1. RA	Rapid to tool change position.
17. END	End Program.

1.	Program a feedrate as a V20. For example, V20 8. will be a feedrate of 8 inches per minute when using the G80 series.
2.	Program a dimension in V21 as the height which you want the tool to start feeding into the workpiece. This will also be the point which the tool will retract to. If G83 or G87 is used, this dimension must be .100 above the surface which will be drilled into. V21 .1 will set the distance at .100 above zero. This dimension must always be an absolute value.
3.	If G82 or G89 is going to be used, program V22 and the time (in seconds) that the tool will dwell at the bottom of the hole. V22 1.5 will cause a 1.5 second dwell at the bottom of each hole when using G82 or G89. If any other G80 series code is used, a dwell is not performed, therefore, V22 does not need to be entered. If entered, V22 must be positive.

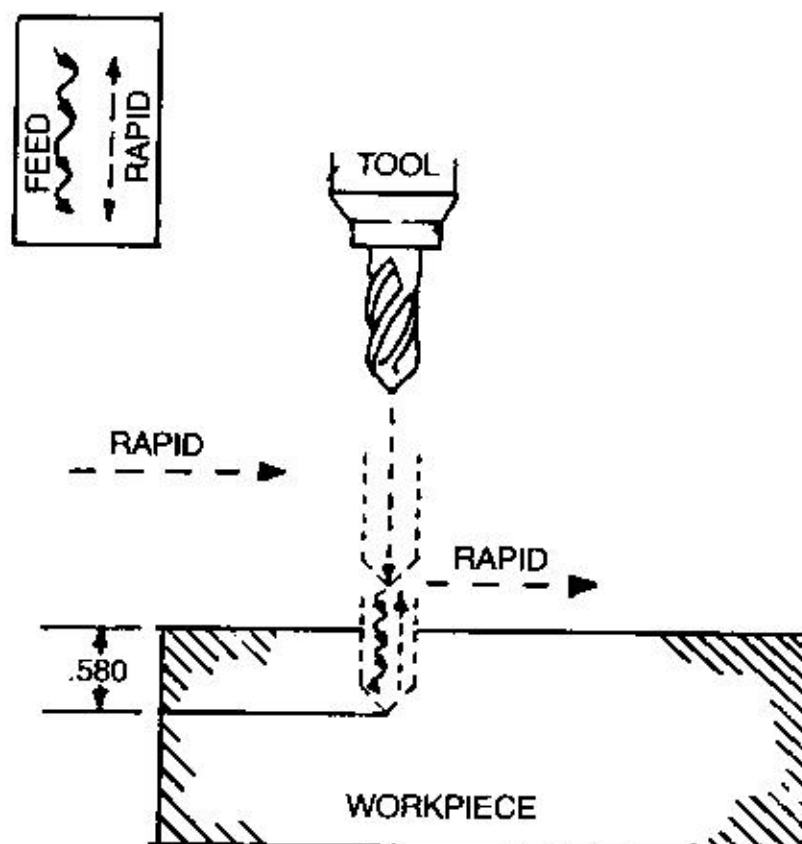
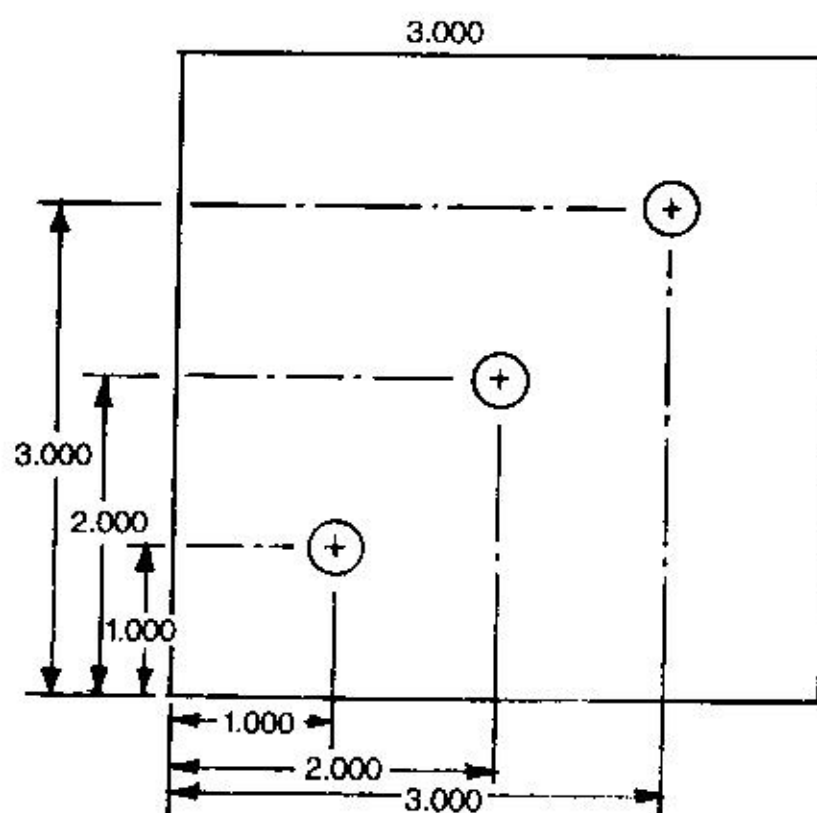


Figure 24



4. If a pecking cycle G83 or G87 is used, the maximum peck is given as V23. The control will make each peck equal and will not exceed the depth given. V23 .160 will set the maximum peck at .160. This distance is given as a positive incremental value.
5. Program the appropriate G code. A listing of all G80 series codes and a description of each is given in Section .
6. Program the X Y coordinates and Z depth of the first hole. This move should be in Rapid. When this event is executed, the control will rapid in X and Y to the programmed position, rapid the Z axis to the point given in V21 as the height to start drilling from. It will then drill the hole and retract to the starting height.

List the X and/or Y positions for the remaining holes to be drilled. Each time a new position is reached, the G code will become active and the hole will be drilled. After the last hole, program a G80 to deactivate the drilling cycle. Using Event Search or changing modes (except changing from Single Step to Auto) will also deactivate any drilling cycle.

When canned cycles are being executed, disregard the event number being displayed. The event numbers displayed are events that the internal software uses.

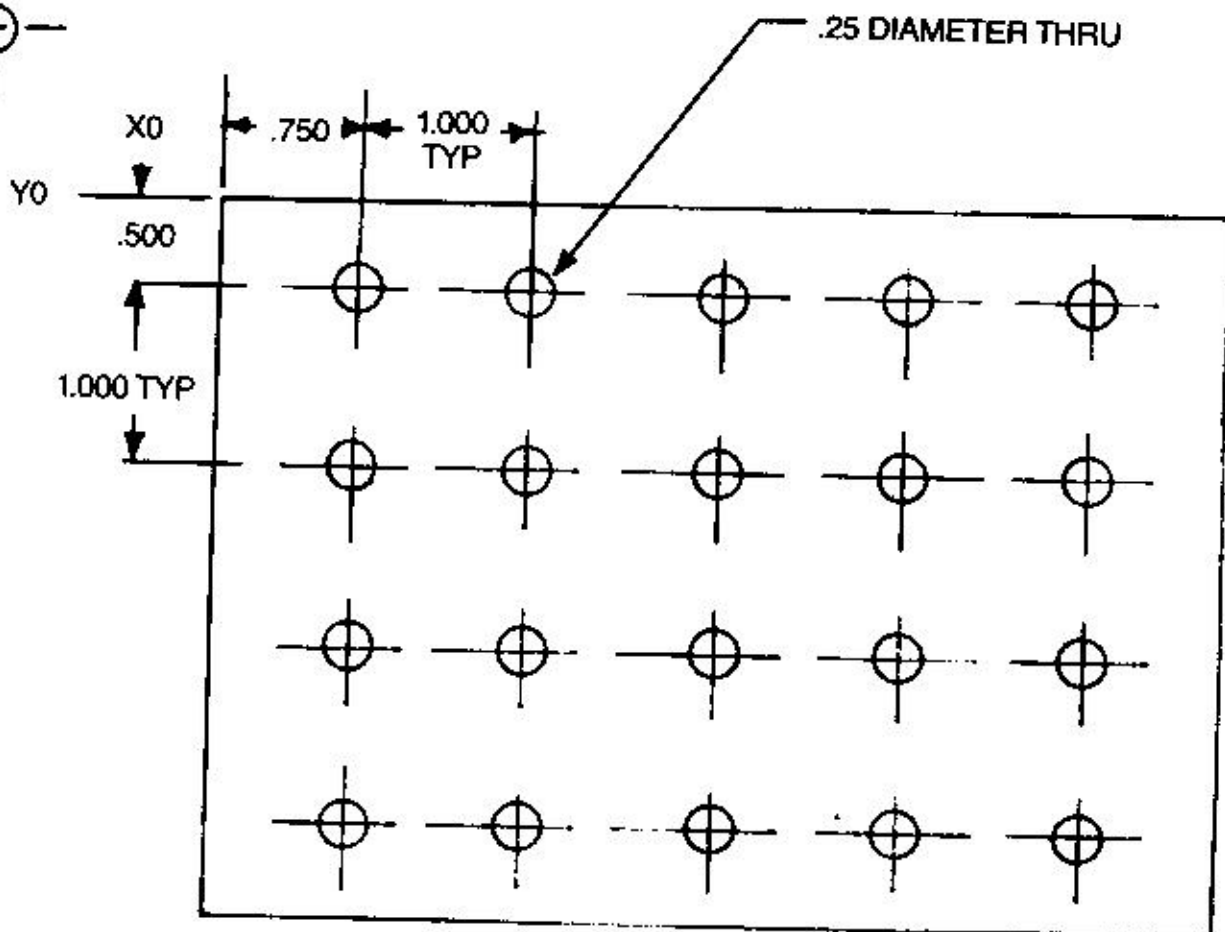
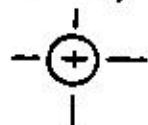
If a feed rate is not entered or your maximum peck is larger than the total depth drilled, the machine will dwell upon execution of the canned cycle. If the machine does this, check your program for an error using Program Check.

When peck drilling is used, the control will assume that the V21 dimension is .100 (2.5mm in metric) above the top of the workpiece. The control then adds this distance to the first peck so that all pecks are equal value and all pecks are into the workpiece.

8. DO LOOPS PROGRAMMING EXAMPLE

Figure 25 shows a part with equally spaced holes. The same procedures outlined in section A should be followed for each part. The first 3 steps again will be: 1) how the part is held, 2) where zero will be set, and 3) what tooling will be used. Since the part is small and rectangular, hold it in a vise. Absolute zero will be set at the upper left corner. Tool 1 will be a spot drill, Tool #2 will be a .250 diameter drill.

$X-2.$
 $Y+1.$ } TOOL CHANGE POSITION



MATERIAL: .375 x 4.000 x 5.500 ALUMINUM

Figure 25

The program will first list the offsets, then retract the quill and move to the tool change position. The first hole will be spot drilled using G81. The remaining 4 holes in the first row will be drilled using a do-loop. After all holes are center-drilled, the G code will be deactivated, the quill retracted and moved to the tool change position. Tool 2 will drill the holes, again using G81 and a series of do-loops. Refer to figure as you go through the program.

<u>EVENT</u>	<u>FUNCTION</u>
1. TOOL 1	Set offset #1.
2.	Open event for value of offset #1.
3. TOOL 1002	Set offset #2.
4.	Open event for value of offset #2.
5. TOOL 0	Deactivate offsets.
6. Z0 RA	Retract quill.
7. X-2. Y1. RA	Tool change position.
8. TOOL 1	Activate offset #1.
9. V20 14.	Set the feedrate at 14. IP%.
10. V21.1	Sets the drilling start height.
11. G81	Activate drilling cycle for spot drill.
12. X.75 Y-.500 Z-.14 RA	The XY location and Z depth of the first hole. The event is programmed in Rapid but because G81 is active, the Z axis will rapid to the start height then feed to the Z axis depth.

13. DO 4	Do loop for the 1st line of 4 holes.
14. X1. RI	Incremental distance between holes (G81 is active to drill each increment). The do loop will cause 4 holes to be spot drilled at 1. increments.
15. END	End for do loop. The program will continue because this event only ends the do loop.
16. Y-1.0 RI	Incremental move to next line (G 81 is still active so first hole is spot drilled.)
17. DO 4	Do loop for next line of holes.
18. X-1.0 R I	Incremental distance in the negative direction.
19. END	End for do loop.
20. Y-1. RI	Incremental move to the third line of holes. (G81 still active so first hole is spot drilled.)
21. DO 4	Do loop for next line of holes.
22. X1.0 R	Incremental distance.
23. END	End of do loop
24. Y-1.0 RI	Incremental move to last line of holes.
25. DO 4	Do loop for last line.
26. X-1.0 RI	Incremental distance in the negative direction.
27. END	End of do loop.
28. G 80	Deactivate drilling cycle.
29. TOOL 0	Deactivate tool offset
30. Z0 RA	Retract drill.
31. X-2. Y1. RA	Tool change position.

32. TOOL 2	Activate offset for tool #2, stops program.
33. V20 10.	Sets feedrate at 10 IPM.
34. V21 .1	Sets drilling start height.
35. G 81	Activate drilling cycle.
36. X.750 Y-.500 Z-.500 RA	Is the X Y location and Z depth of the first hole.
37. DO 4	Do loop.
38. X1.0 R 1	Increment. (G81 active drills 4 holes at 1. increments.)
39. END	END for do loop.
40. Y-1. R1	Incremental move and drills first hole in next run.
41. DO 4	Do loop.
42. X-1.0 R1	Increment.
43. END	End do loop.
44. Y-1.0 R1	Incremental move down.
45. DO 4	Do loop.
46. X1.0 R 1	Increment.
47. END	End do loop.
48. Y-1.0 R 1	Incremental move down.
49. DO 4	Do loop.
50. X-1.0 R 1	Increment.
51. END	End do loop.
52. G 80	Deactivate drilling cycle.
53. TOOL 0	Deactivate tool offset.
54. Z0 RA	Retract drill.
55. X-2. Y1. RA	Tool change position.
56. END	End of program.

Enter this program and see how the do loops work with the drilling cycle. It will be good practice.

C. SUBROUTINES PROGRAMMING EXAMPLE

Figure 26 shows how to apply a subroutine to the same program. In the last program only 2 tools were used. With this program we will use the 3 tools that are described on the tool sheet. However, by grouping do loops in a subroutine the program will be shorter than the last example even though you added another tool.

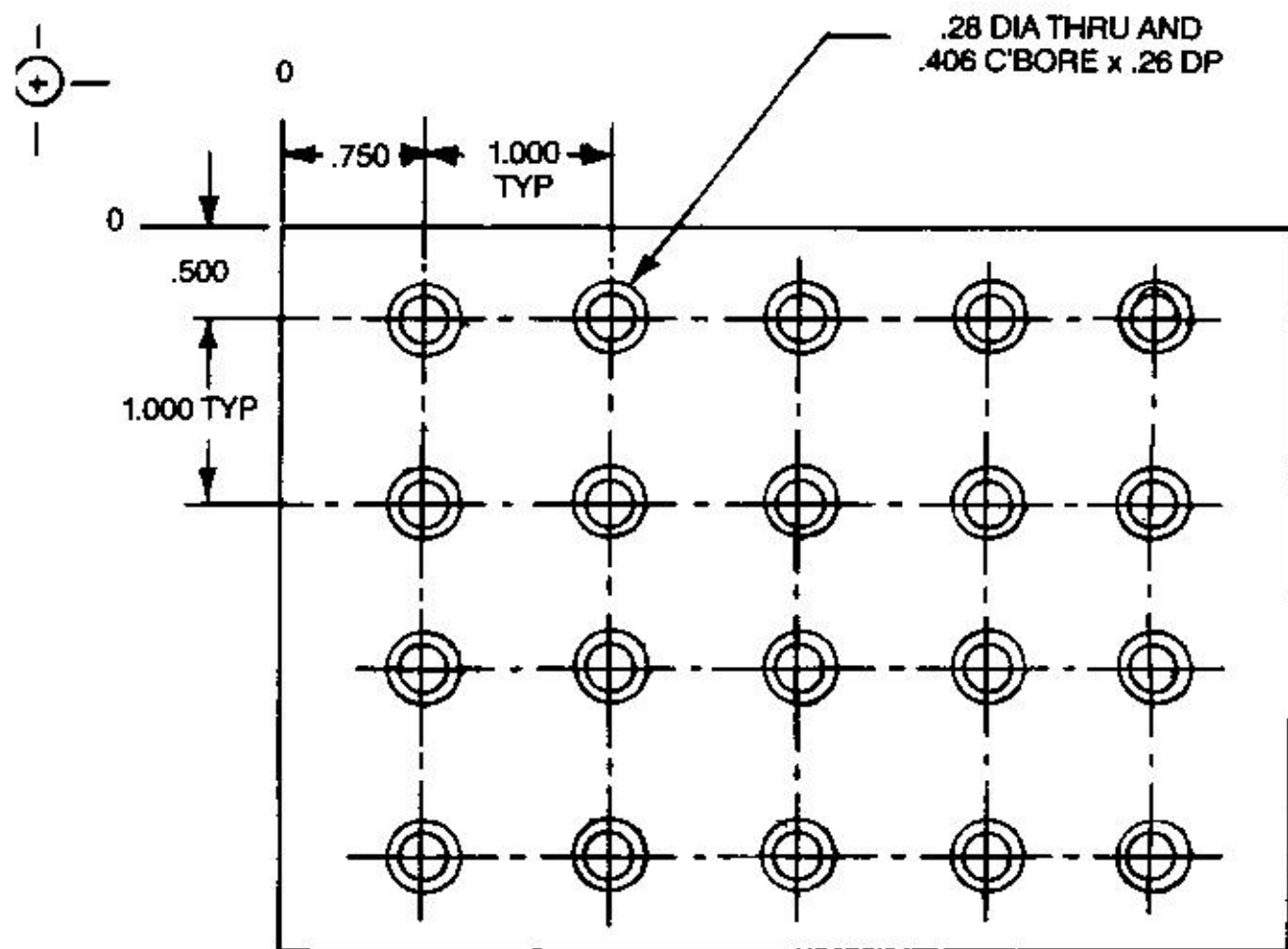
Subroutine 1 will be the series of do loops used to position the tools to the hole locations. The same subroutine will be used to spot drill, drill and counterbore the holes. Use figure for reference as you examine the program.

When writing a subroutine ensure the information in the Sub has only the things you want repeated. As an example, this Sub will be used with all the tools. Each tool goes to the same location but each will have a different feedrate, will use a different canned cycle and will machine to a different depth. So, these are things that you would not put into the subroutine.

The things that will be repeated are: the do loops for each line of holes; the incremental move to each line of holes; and cancellation of the canned cycle when all the positioning moves are complete. No matter what tool you are using, you will need to deactivate the tool offset, retract the quill and move to the tool change position. Because these instructions will be repeated with each tool, these instructions will be in the sub.

Since you already wrote the program without Sub's, turn back to Example 2 and look at events 13 through 31 and 37 through 55. They are exactly the same. That is what the subroutine will look like.

One more thing to remember. A subroutine is always entered into the events that follow the main program. If the main program is from event 1 through event 29, the subroutine would be entered at event 30. To search for a subroutine when editing, press
SUBR 1 EVENT SEARCH.



MATERIAL: .375 x 4.000 x 5.500 ALUMINUM

Figure 26: SUBROUTINES

1. TOOL 1001	Set offset #1.
2.	Open event for value of offset #1.
3. TOOL 1002	Set offset #2.
4.	Open event for value of offset #2.
5. TOOL 1003	Set offset #3.
6.	Open event for value of offset # 3.
7. TOOL 0	Deactivate any tool offset.
8. Z0 RA	Retract quill.
9. X-2. Y 1. RA	Move to tool change position.
10. TOOL 1	Activate offset #1.
11. V20 15.	Sets spot drilling rate at 15 IPM.
12. V21.1	Set drilling start height.
13. G81	Activates canned drilling cycle.
14. Z-.140 RA	Depth to be spot drilled. The G81 drilling cycle will not take effect until an X and/or Y coordinate is executed.
15. CALL 1	Call subroutine #1. This will spot drill all holes because G-81 is active. Go to SUB 1 and follow the program to see how the holes will be spot drilled. After the sub is executed, the program will return to Event 16.
16. TOOL 2	Activate offset #2.
17. V20 10.	Sets drilling feedrate at 10 IPM.
18. V21.1	Sets drilling start height.
19. G81	Activates canned drilling cycle.
20. Z-.5 RA	Depth to be drilled.

21. CALL 1	Call suproutine #1. This drills all holes because G81 is active.
22. TOOL 3	Activate offset #3.
23. V20 8.	Sets counterboring feedrate at 8 IPM.
24. V21.1	Sets counterboring start depth.
25. V22 2.	Sets dwell time at 2 seconds at bottom of counterbore.
26. G 82	Canned counter boring cycle.
27. .Z-.260 RA	Depth to be counter bored.
28. Call 1	Call subrouting #1. This will counter-bore all holes because G82 is active.
29. END	End of program.
30. SUB 1	Label this pattern as subroutine 1.
31. .X.750 Y-.500 RA	First X Y location in pattern.
32. DO 4	Do loop for 4 holes.
33. Y-1.0 RI	Incremental distance in Y axis to next line of holes.
34. END	END for do loop.
35. Y-1.0 RI	Incremental distance in Y axis to next line of holes.

ANILAM CRUSADER PROGRAM SHEET

SHEET 1 OF 2

PART OR DRWG. # EXAMPLE 3: SUBROUTINES PROGRAMMER M W A

DATE 9-16-85

TOOL LIST

OFFSETS

TOOL 1: SPOT DRILL @ 3,500 R.P.M.
TOOL 2: .28 DRILL @ 3,000 R.P.M.
TOOL 3: .406 ENDMILL @ 2,500 R.P.M.

SETUP INSTRUCTIONS

X, Y "Ø" SET IN UPPER LEFT CORNER.
PART IS HELD IN VICE WITH STOP ON LEFT SIDE

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1	TOOL	1001				21	CALL	1			
2	X Ø		Z T.L.O	R	A	22	TOOL	3			
3	TOOL	1002				23	V20	8.			
4	X-28		Z T.L.O	R	A	24	V21	.1			
5	TOOL	1003				25	V22	2.			
6	X-406		Z T.L.O	R	A	26	GØ2				
7	TOOL	Ø				27					
8			Z Ø	R	A	28	CALL	1		Z-.26	R A
9	Y-2.	Y 1.		R	A	29	END				
10	TOOL	1				30	SUB	1			
11	V20	15.				31	X-.750	Y-.5		R	A
12	V21	.1				32	Do	4			
13	GØ1					33	X 1.			R	I
14			Z-.14	R	A	34	END				
15	CALL	1				35		Y-1.		R	I
16	TOOL	2				36	Do	4			
17	V20	10.				37	Y-1.			R	I
18	V21	.1				38	END				
19	GØ1					39		Y-1.		R	I
20			Z-.5	R	A	40	Do	4			



ANILAM CRUSADER PROGRAM SHEET

RT OR DRWG. # _____ PROGRAMMER _____ DATE _____

TOOL LIST

OFFSETS

SETUP INSTRUCTIONS

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
41	X 1.			R	I	1					
42	END					2					
43		Y-1.		R	I	3					
44	Do	4				4					
45	X-1.			R	I	5					
46	END					6					
47	G80					7					
48	TOOL	ϕ				8					
49			Z ϕ	R	A	9					
50	X-2.	Y1.		R	A	0					
51	END					1					
52						2					
53						3					
54						4					
55						5					
56						6					
57						7					
58						8					
59						9					
60						0					



36. DO 4	Do loop for 4 holes.
37. X-1.0 RI	Negative incremental distance across X axis between holes.
38. END	End for do loop.
39. Y-1.0 RI	Incremental distance in Y axis to next line of holes.
40. DO 4	Do loop for 4 holes.
41. X1.0 RI	Incremental distance in X axis to next line of holes.
42. END	End for do loop.
43. Y-1.0 RI	Incremental distance in Y axis to next line of holes.
44. DO 4	Do loop for 4 holes.
45. X-1.0 RI	Negative incremental distance across X axis between holes.
46. END	End for do loop.
47. G. 80	Deactivate canned drilling cycle.
48. TOOL 0	Deactivate tool offset.
49. Z0 RA	Retract drill.
50. X-2. Y1. RA	Move to tool change position.
51. END	End of subroutine.

By using subroutines, you cut the program from 56 events (using 2 tools) to 51 events (using 3 tools.) You saved 5 events and did one completely new operation.

D. NESTING PROGRAMMING EXAMPLE

In this program, we used "nesting". Nesting is putting a pattern inside of another pattern. Our do loops were the first patterns. But by grouping them in a subroutine, we put our first pattern into a sub pattern that can be called anytime we need it.

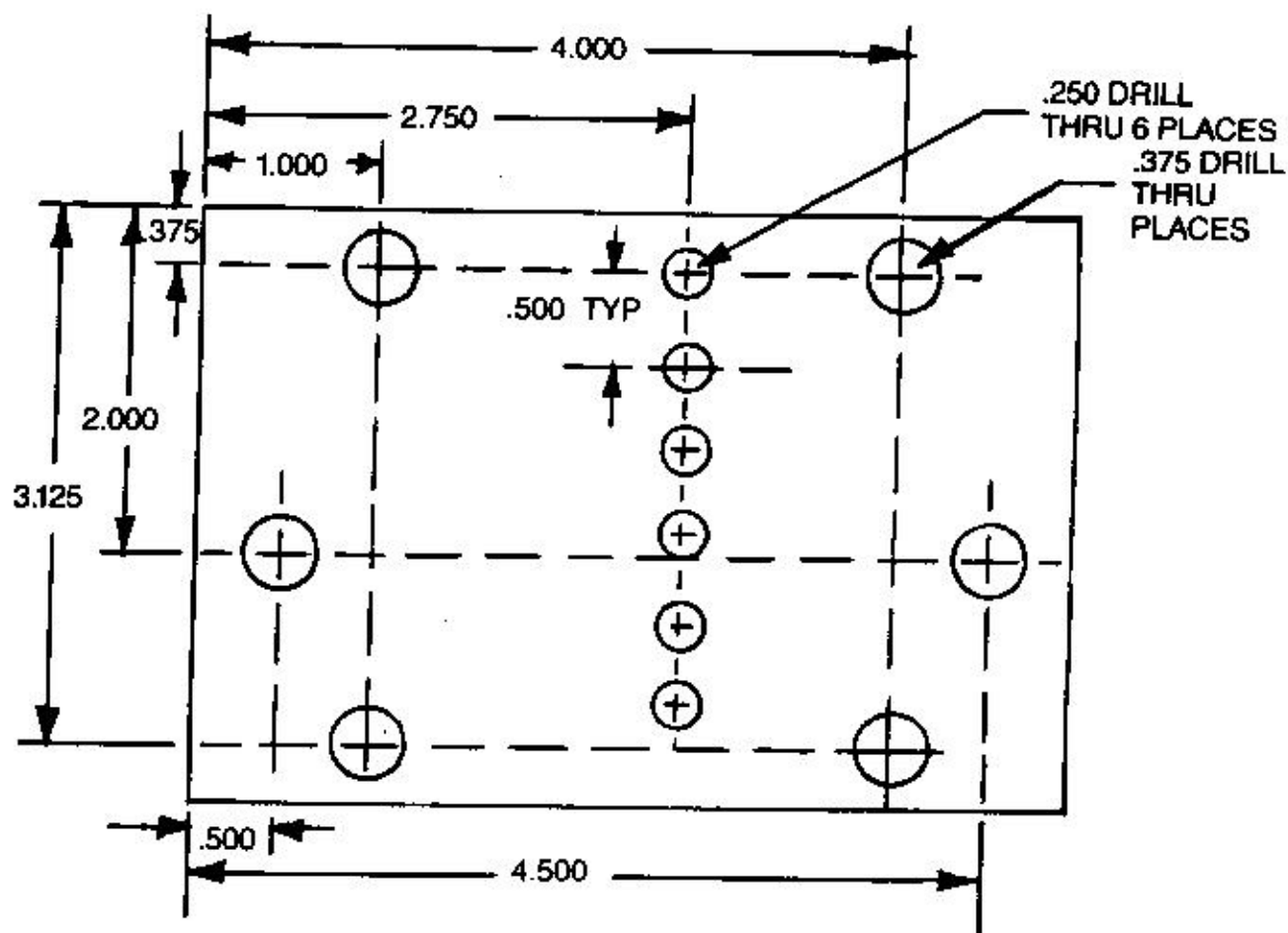
Putting do loops into subroutines is one form of nesting. Another form is to call a subroutine and in that subroutine call a second subroutine. There is no limit now for you can nest with the Crusader.

E. PECK DRILLING PROGRAMMING EXAMPLE

In this example, you will use the G83 (Peck drilling) code to make the part shown in figure 27. The middle row of holes will be listed in subroutine 1. The outside pattern will be listed as subroutine 2.

A spot-drill (tool 1) will use both subroutines and will spot-drill each hole using the G81 code. The second tool, a .250 drill (tool 2), will use subroutine 1 and will peck drill the center row of holes using G83. A .375 drill (tool 3) will use subroutine 2 and will peck drill the outside pattern of holes using the G83 code. An explanation of each event for this program is listed below:

- | | |
|----------------|--|
| 1. TOOL 1001 | Set offset #1. |
| 2. | Open event for value of offset #1. |
| 3. TOOL 1002 | Set offset #2. |
| 4. | Open event for value of offset #2. |
| 5. TOOL 1003 | Set offset #3. |
| 6. | Open event for value of offset #3. |
| 7. TOOL 0 | Deactivates offset. |
| 8. Z0 RA | Retracts quill. |
| 9. X-2. Y1. RA | Rapids to tool change position. |
| 10. TOOL 1 | Activate tool 1's offset, stops program. |
| 11. V20. 15. | Sets drilling feedrate. |
| 12. V21 .1 | Sets starting height. |



MATERIAL: 1.000 x 3.500 x 5.000 ALUMINUM

Figure 27: PECK DRILLING

13. G 81	Activates canned cycle.
14. Z-.2 RA	Drilling depth.
15. CALL 2	Executes SUB 2, spot drill outside hole pattern.
16. Z-.135	Sets new drill depth for middle row of holes.
17. CALL 1	Execute sub 1, spot drill middle row of holes, cancels drilling, cancels tool offset, moves to tool change.
18. TOOL 2	Activates tool 2 offset, stops program.
19. V20 8.	Sets feedrate at 8 IPM.
20. V21 .1	Sets starting height.
21. V21 .1	Sets maximum peck at .15 deep. The control will calculate how deep to make each peck so that each one is an equal depth and does not exceed the total depth.
22. G 83	Activates canned peck drilling cycle. The tool will retract to the starting height after each peck.
23. Z-1.15 RA	Sets drilling depth.
24. CALL 1	Executes sub 1 which peck drills middle row of holes, deactivates G83, deactivates tool length offset, retracts quill and moves to tool change position.
25. TOOL 3	Activates tool 3 offset, stops program.
26. V20 10.	Sets feedrate at 10 IPM.
27. V21 .1	Sets starting height at .1.
28. V23 .2	Sets max peck at .2.
29. G83	Activates canned peck drilling.

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 4: PEEK DRILLING PROGRAMMER DIFF DATE 11-9-81

TOOL LIST						OFFSETS		SETUP INSTRUCTIONS					
Tool 1: SPOT DRILL @ 3,500 RPM								PART HELD IN VICE WITH STOP ON LEFT SIDE. X, Y Ø AT TOP LEFT CORNER. PARALLELS 5/32" THICK MAY.					
Tool 2: .250 DRILL @ 3,000 RPM													
Tool 3: .375 DRILL @ 2,500 RPM													
EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS		
1	Tool	1001				21	V23	.1					
2	X Ø		Z TLO	R	A	22	G83						
3	Tool	1002				23			Z-1.15	R	A		
4	X-.25		Z TLO	R	A	24	CALL	1					
5	Tool	1003				25	Tool	3					
6	X-.375		Z TLO	R	A	26	V20	10.					
7	Tool	Ø				27	V21	.1					
8			Z Ø	R	A	28	V23	.2					
9	X-.2	Y1.		R	A	29	G83						
10	Tool	1				30			Z-1.2	R	A		
11	V20	15.				31	CALL	2					
12	V21	.1				32	G80						
13	G81					33	Tool	Ø					
14		Z-.2		R	A	34			Z 0	R	A		
15	CALL	2				35	X-.2	Y1.		R	A		
16		Z-1.25		R	A	36	END						
17	CALL	1				37	END 1						
18	Tool	2				38	X2.75	Y-.375		R	A		
19	V20	8.				39	DO	5					
20	V21	.1				40		Y-.500		R	A		

A

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 4: POCK DRILLING PROGRAMMER DFF DATE 11-9-85

TOOL LIST

OFFSETS

SETUP INSTRUCTIONS

SAME AS SHEET 1

SAME AS SHEET 1

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1	END					1					
2	G80					2					
3	TOOL	Ø				3					
4			Z Ø	R	A	4					
5	X-2.	Y 1.		R	A	5					
6	END					6					
7	SUB	2				7					
8	X1.	Y-375		R	A	8					
9	X .5	Y-2.		R	A	9					
0	X1.	Y-3.125		R	A	0					
1	X4.			R	A	1					
2	X4.5	Y-2.		R	A	2					
3	X4.	Y-375		R	A	3					
4	END					4					
5						5					
6						6					
7						7					
8						8					
9						9					
0						0					

30. Z-1.2 RA	Sets drilling depth.
31. CALL 2	Executes SUB which peck drills outside pattern of holes.
32. G80	Deactivates G 83.
33. TOOL 0	Deactivates tool length offset.
34. Z0 RA	Retracts quill.
35. X-2. Y1. RA	Rapids to tool change location.
36. END	End of program.
37. SUB 1	Start of subroutine 1.
38. X2.750 Y-.375 RA	Moves to top hole of middle row.
39. DO 5	Sets do loop to be repeated 5 times.
40. Y-.500 RI	Incremental distance between holes.
41. END	END of do loop.
42. G 80	Deactivates canned drilling cycle.
43. TOOL 0	Deactivates tool length off set.
44. Z0 RA	Retracts quill.
45. X-2. Y1. RA	Rapids to tool change position.
46. END	End of subroutine 1.
47. SUB 2	Start of subroutine 2.
48. X1. Y-.375 RA	Events 48-53 list X Y locations of the one quadrant pattern of holes.
49. X.500 Y-2. RA	
50. X1. Y-3. 125 RA	
51. X4. RA	
52. X4.5 Y-2. RA	
53. X4. Y-.375 RA	
54. END	End of subroutine 2.

F. MIRROR IMAGE PROGRAMMING EXAMPLE

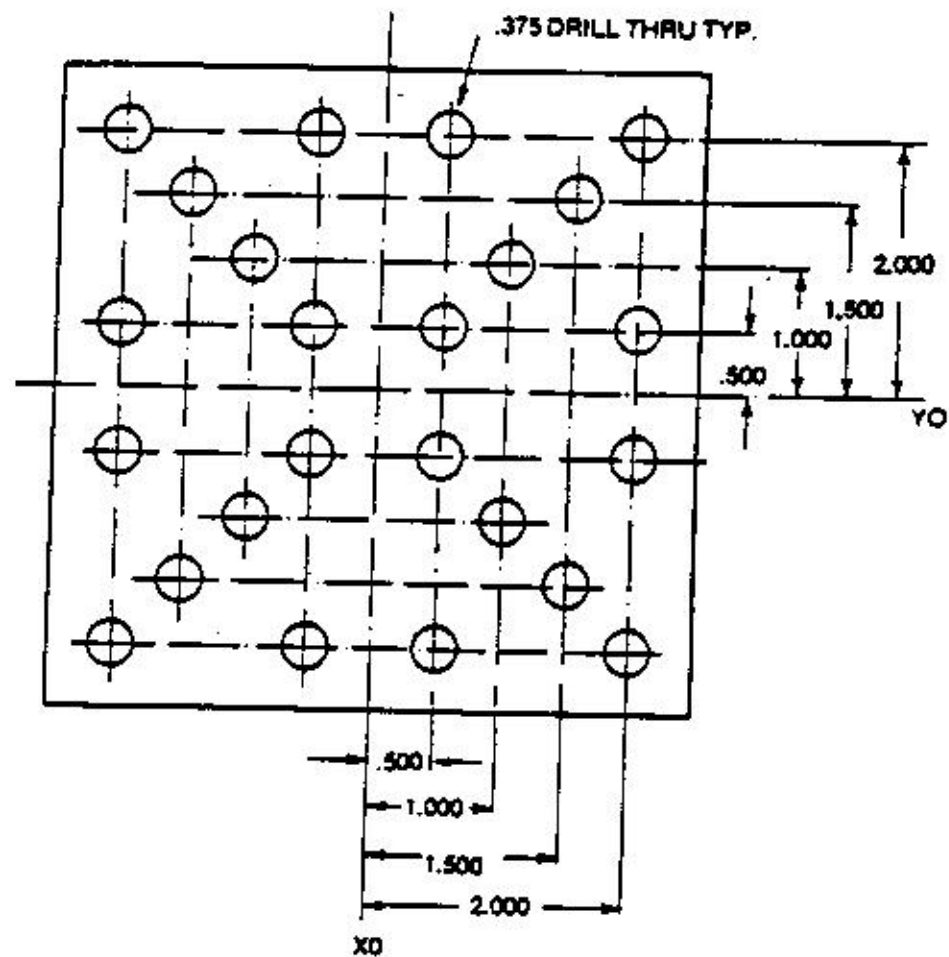
Mirror image is used to change sign of a dimension, such as changing X+ to X-. This feature is helpful for many applications such as left and right handed parts or mold making. The part shown for Example #5 (Figure 28) has a hole pattern that is the same on each side of the zero reference point. By writing a subroutine with the hole locations for one quarter, that pattern can be flipped using Aux codes X and Y (together) from plus to minus or minus to plus.

The program for Example 5 will be written using 2 subroutines, the first will have all the hole locations in the upper right quadrant. This quadrant is chosen because all the dimensions are X+ Y+ and you do not have to remember to put a minus sign in.

SUB 2 will call SUB 1 which will position the tool to all the locations in the upper right quadrant. SUB 2 will position the tool to all the locations in the upper right quadrant. SUB 2 will then activate Aux 100 to mirror all the X+ dimensions to X- and call SUB 1 again. This will drill the holes in the X- Y+ quadrant. AUX 200 is then to be programmed to mirror the pattern in the Y axis. Again, SUB 1 will be called. This will drill the holes in the X+ Y- quadrant. Next, we must mirror the original pattern across X and Y and again call SUB 1. This drills the holes in the X- Y- quadrant.

At AUX 800 is programmed to put our signs (+ or -) back to normal. The G81 and offset are deactivated, the drill is retracted and sent to tool change position. In this way we can use SUB 2 to spot drill then drill the entire part.

- | | |
|-----------------|---|
| 1. TOOL 1001 | Sets offset #1. |
| 2. | Open event for value of offset 1. |
| 3. TOOL 1002 | Sets offset #2. |
| 4. | Open event for value of offset. |
| 5. TOOL 0 | Deactivates tool offsets. |
| 6. Z0 RA | Retracts drill. |
| 7. X-3.5 Y0. RA | Rapid to tool change position. |
| 8. TOOL 1 | Activates tool 1 offset, program stops. |
| 9. V80 15. | Sets drilling feedrate. |



MATERIAL: .500 x 5.000 x 5.000 ALUMINUM

Figure 28

10. V21 .1	Sets drilling start height.
11. G81	Activates drilling canned cycle.
12. Z-.135	Sets drilling depth.
13. CALL 2	Executes SUB 2 which mirrors SUB 1 to drill entire part. See SUB 2 for details.
14. TOOL 2	Activates tool 2 offset.
15. V20 15.	Sets feedrate at 15 IPM.
16. V21 .1	Sets drilling - start height.
17. V23 .2	Sets maximum peck distance.
18. G83	Activates peck drill cycle.
19. Z-.65 RA	Sets drilling depth.
20. CALL 2	Executes SUB 2 to peck drill part.
21. END	End of program.
22. Sub 1	Start of sub 1.
23. X.500 Y 3. RA	Events 23-28 list the X and/or Y locations for the holes in the X+ Y+ quadrant.
24. X2. RA	
25. X1.500 Y.500 RA	
26. X1. Y1. RA	
27. X.500 Y.500 RA	
28. X2. RA	
29. END	End of SUB 1.
30. Sub 2	Start of SUB 2.
31. CALL 1	Executes SUB 1 which drills holes in X+Y+ quadrant.
32. AUX 100	Instructs the control to change the sign of X axis dimensions from here on.

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 5: MIRROR IMAGE PROGRAMMER BJT DATE 2-13-85

TOOL LIST		OFFSETS		SETUP INSTRUCTIONS							
TOOL 1: SPOT DRILL @ 3,000 RPM				PAR - HOLE IN VISE WITH 1" ON LEFT SIDE							
TOOL 2: .375 DRILL @ 2,500 RPM				X, Y ϕ IS AT THE CENTER							
				PAR -							
EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1	TOOL	1001				21	END				
2	X 0		Z TLO	R	A	22	SUB	1			
3	TOOL	1002				23	X.5	Y 2.		R	A
4	X.375		Z TLO	R	A	24	X 2.			R	A
5	TOOL	ϕ				25	X 1.5	Y.5		R	A
6			Z ϕ	R	A	26	X 1.	Y 1.		R	A
7	X-3.5	Y ϕ		R	A	27	X.5	Y.5		R	A
8	TOOL	1				28	Y 2.			R	A
9	V20	15.				29	END				
10	V21	.1				30	SUB	2			
11	G81					31	CALL	1			
12			Z-13.5	R	A	32	AUX	100			
13	CALL	2				33	CALL	1			
14	TOOL	2				34	AUX	200			
15	V20	15.				35	CALL	1			
16	V21	.1				36	AUX	300			
17	V23	.2				37	CALL	1			
18	G82					38	AUX	800			
19			Z-6.5	R	A	39	G80				
20	CALL	2				40	TOOL	ϕ			



ANILAM CRUSADER PROGRAM SHEETPART OR DRWG. # EXAMPLE 5: HPPROX IMAGE PROGRAMMER B J T DATE 2-13-85

TOOL LIST

OFFSETS

SETUP INSTRUCTIONS

SAME AS SHEET 1

SAME AS SHEET 1

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1			Z Ø	R	A	1					
2	X-3.5	Y Ø		R	A	2					
3	END					3					
4						4					
5						5					
6						6					
7						7					
8						8					
9						9					
10						0					
11						1					
12						2					
13						3					
14						4					
15						5					
16						6					
17						7					
18						8					
19						9					
20						0					



33. CALL 1	Executes SUB 1 with AUX 100 active, drills holes in X- Y+ quadrant.
34. AUX 200	Changes the sign of Y axis commands.
35. CALL 1	Executes SUB 1 with AUX 200 active. Drills holes in X+Y- quadrant.
36. AUX 300	Changes the signs of X- and Y- commands.
37. CALL 1	Executes sub 1 with Aux 300 active. Drills holes in X- Y- quadrant.
38. AUX 800	Deactivates mirror image.
39. G80	Deactivates drilling cycle.
40. TOOL D	Deactivates tool length offset.
41. ZD RA	Retracts quill.
42. X-3.500 YD RA	Rapids to tool change position.
43. END	End of SUB 2.

SECTION 9
MILLING EXAMPLES

SECTION 9: MILLING EXAMPLES

It is necessary to understand the format rules that are used with the Crusader Series-M. These rules were explained earlier in section 8-1. Please review that information prior to reading the following examples.

A. CIRCULAR MILLING PROGRAMMING EXAMPLE

Tool #1 is a .500 end mill. It will be used at 1800 RPM with a feedrate of 10 IPM. The part is screwed to a fixture plate, X and Y zero is set in the lower left hand corner of the part.

NOTE:

Remember that you are programming the center of the tool so the programmed radius of each corner will be the part radius + or - the cutter radius.

You will use the upper left hand corner for the start point and will climb mill around the part. The tool will be positioned so the cutting edge is on the top edge of the part and .100 off the left side of the part. (This is for clearance so the tool does not crash into the side of the workpiece.) This is Milling Position 1.

Each milling position is numbered in the order in which you will machine the part. Review figure 29 and use it for reference as you write the program.

The set-up and tool list are put at the top of the programming sheet. Events 1-6 are the standard format for starting a program. The offsets are listed, the quill is retracted and moved to the tool change position. Tool 1's offset is activated.

Event 7: Moves the tool in rapid to position 1.

Event 8: Rapids the tool to .100 above the top of the part.

Event 9: Sets the feedrate to 10.0 IPM.

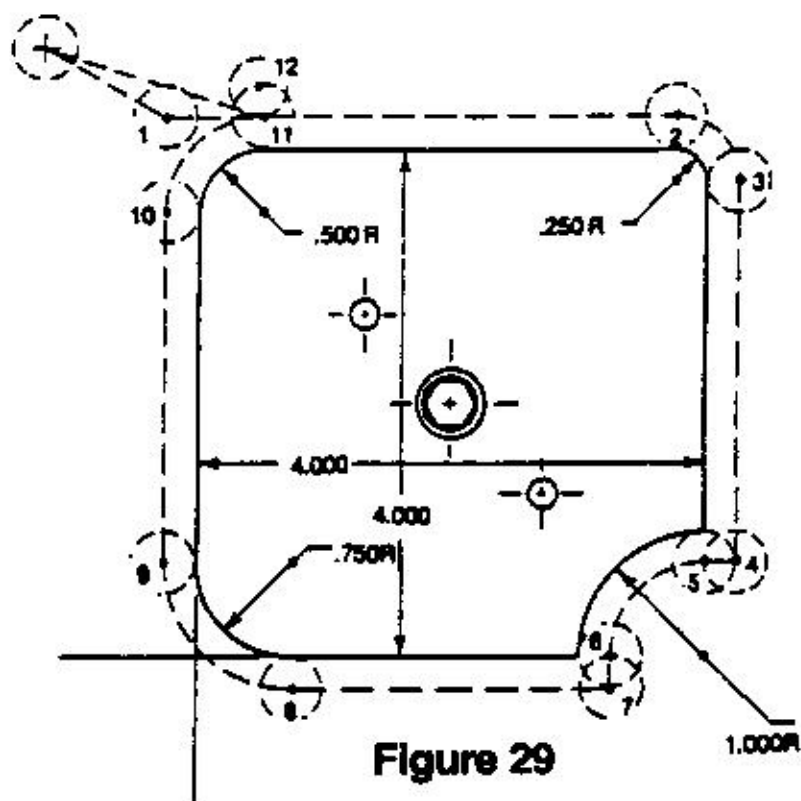
Event 10: Feeds the tool to cutting depth.

Event 11: Feeds the tool to position 2 (rules 1 and 2 for circular milling from Section).

Event 12: Describes arc and direction CW (rule 3).

Event 13: Describes the arc center (rule 4).

X-2
Y-5. ABSOLUTE TOOL CHANGE POSITION



ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # FRANCIS G. GREGG PROGRAMMER J T W DATE 10-31-9

TOOL LIST						OFFSETS						SETUP INSTRUCTIONS					
Tool 1: .5 END MILL @ .001												30000 RPM TO CENTER Set X Y Z TO .000 CENTER OF PART					
EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS					RAPID / FEED	INCR / ABS
1	Tool	10.0				21	Arc	CCW									
2	X.5		Z+.0	R	A	22		Y-.25		F	A						
3	Tool	Ø				23	X.750			F	A						
4			ZØ	R	A	24	Arc	CW									
5	X-2.	Y5.		R	A	25	X.750	Y.75		F	A						
6	Tool	1				26	X-.25	Y.75		F	A						
7	Y-.350	Y4.25		R	F	27	Arc	CW									
8			Z.1	R	A	28		Y3.5		F	A						
9	FEED	10.				29	Arc	CW									
10			Z-.5	F	A	30	X.5	Y3.5		F	A						
11	Y3.75			F	A	31	X.5	Y4.25		F	A						
12	Arc	CW				32	Arc	CW									
13	Y3.75	Y3.75		F	A	33		Y.1		F	I						
14	Y4.25	Y3.75		F	A	34	Tool	Ø									
15	Arc	CW				35			ZØ	F	A						
16		Y.75		F	A	36	X-2.	Y5.		F	A						
17	X4.			F	A	37	END										
18	Arc	CCW				38											
19	X Ø	Y-.750		F	I	39											
20	X-.750	Y-.750		F	I	40											

Event 14: Describes the finish point of the arc (rule 5).

Event 15: The last arc event (rule 6).

Event 16: Feeds the tool to position 4.

Event 17: Feeds the tool to position 5. This is needed to move the tool to the start point of the arc (rule 1 and 2.)

Event 18: Describes arc and direction CCW (rule 3).

Event 19: Describes the arc center. It is described incrementally from the start point. The center point and end point must be defined as a 2 axis position (rule 4).

Event 20: Describes the finish point of the arc (rule 5).

Event 21: The last arc event.

Event 22: Feeds the tool to position 7.

Event 23: Feeds the tool to position 8.

Event 24: Describes arc and direction CW.

Event 25: Describes the arc center.

Event 26: Describes the arc finish point.

Event 27: The last arc event.

Event 28: Feeds the tool to position 10.

Event 29: Describes arc and direction CW.

Event 30: Describes the arc center.

Event 31: Describes the arc finish point.

Event 32: The last arc event.

Event 33: An incremental move to get the tool off the workpiece.

Event 34: Deactivates tool 1's offset.

Event 35: Retracts the quill.

Event 36: Rapids the tool back to the tool change position.

Event 37: Ends the program.

Enter and check this program into memory. You can put a sheet of paper on the table and a pen in the spindle to draw the tool path to check it. To do this, set the tool offset when the pen is on the paper and change event 10 to ZOFA so you do not drive the pen .550 into the paper.

B. MIRROR_IMAGE_PROGRAMMING_EXAMPLE

This example (Figure 30) has four slots .500 wide that go through the part. A .500 end mill will be used. Since the slots are the same width and length, the dimensions of the slot will be put incrementally into a subroutine and mirror image will be used to invert the direction that the slot moves.

In this subroutine, along with the moves to cut the slot, you will also put the moves that will feed the cutter into the workpiece. When the slot has been cut, this will also move the cutter up out of the piece. Finally, you will put in an PUX 800 to deactivate any other mirror image.

Refer to figure 5 at beginning of this manual for a diagram showing the mirror-image quadrants.

Event 1-6: These events have the same format as all the programs.

Event 7: Moves the cutter to the absolute position of the starting point for the first slot.

Event 8: Positions the cutter .100 above the part.

Event 9: Calls subroutine #1 to machine the slot.

Event 10: Positions the cutter to the absolute position of the starting point for the next slot.

Event 11: Mirrors the X-axis dimensions.

Event 12: Calls subroutine #1.

Event 13: Positions to the start point of the next slot.

ANILAM CRUSADER PROGRAM SHEETPART OR DRWG. # EXHAUST 7: MIRROR IMAGE PROGRAMMER BFD DATE 11-16-84**TOOL LIST****OFFSETS**

TOOL 1: .5 BOTTOM CUTTING HANDHILL
@ 1,600 R.P.M

SETUP INSTRUCTIONS

PLACE PART IN VISE WITH LOCATOR
ON THE LEFT SIDE
SET X, Y ϕ IN UPPER LEFT CORNER
OF PART

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1	TOOL	1001				21	X-2.	Y 1.		R	A
2	X.5		Z TLO	R	A	22	END				
3	TOOL	ϕ				23	SUB	1			
4			Z0	R	A	24	FEED	7.5.			
5	X-2.	Y 1.		R	A	25			Z-3	F	A
6	TOOL	1				26	X-1.25	Y-1.25		F	I
7	X 1.625	Y-3.75		R	A	27			Z-1	F	A
8			Z-1	R	A	28	Aux	800			
9	CALL	1				29	END				
10	X 2.375	Y-3.75		R	A	30					
11	Aux	100				31					
12	CALL	1				32					
13	X 2.375	Y-3.625		R	A	33					
14	Aux	300				34					
15	CALL	1				35					
16	X 1.625	Y-3.625		R	A	36					
17	Aux	200				37					
18	CALL	1				38					
19	TOOL	ϕ				39					
20			Z0	R	A	40					



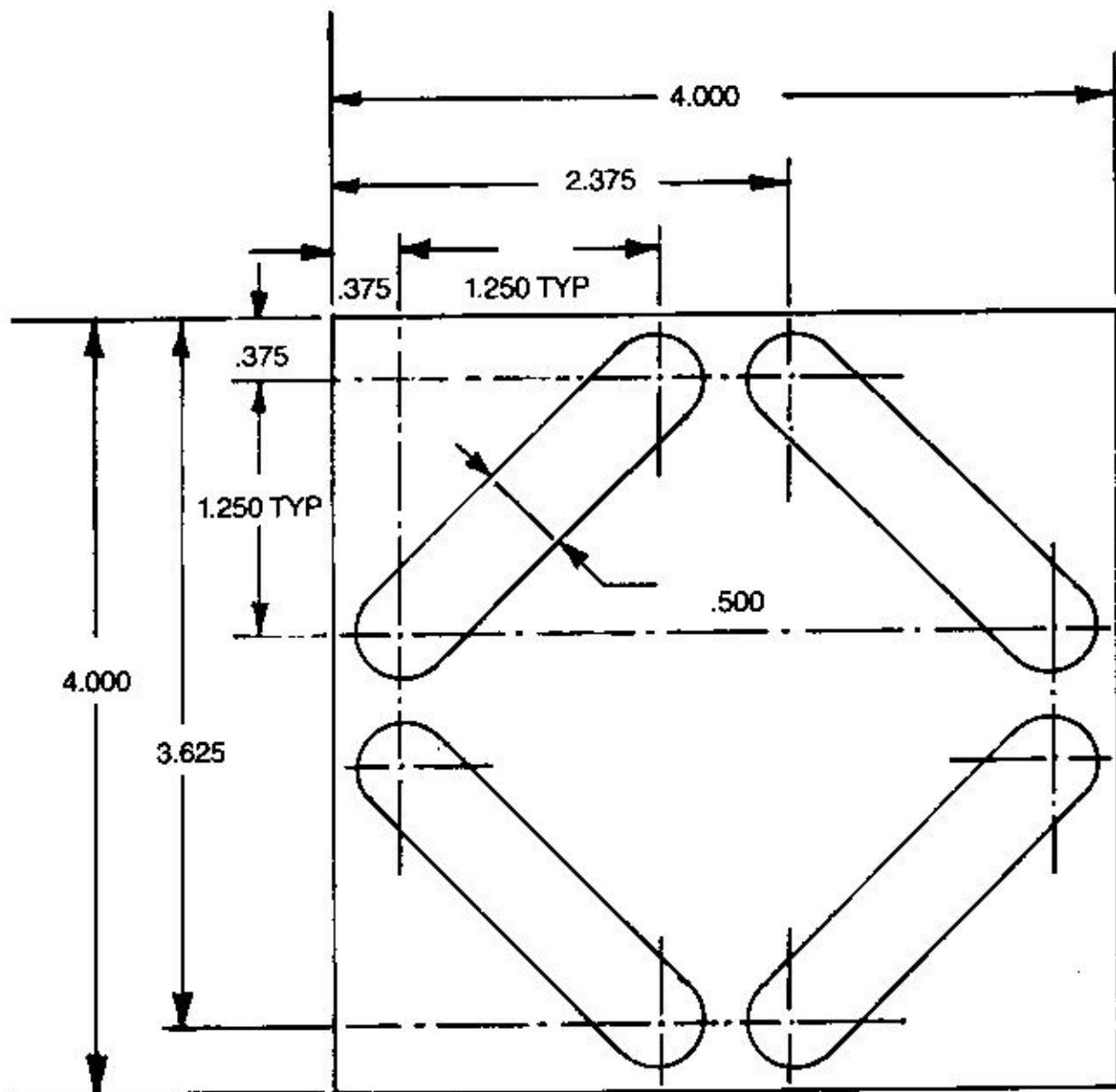


Figure 30: MIRROR IMAGE MILLING

Event 14: Mirrors the X and Y axis dimensions.

Event 15: Calls subroutine #1.

Event 16: Positions to the start point of the last slot.

Event 17: Mirrors the Y axis dimensions.

Event 18: Calls subroutine #1.

Event 19-
22: Deactivates tool offset, retracts quill, moves to the tool change position and ends the program.

Event 23: Labels subroutine #1.

Event 24: Feedrate.

Event 25: Feeds tool into the part.

Event 26: X and Y move to cut the slot.

Event 27: Feeds tool out of the part.

Event 28: AUX code to turn off All mirror images.

Event 29: Ends the subroutine.

C. X and Z CONTOURING PROGRAMMING EXAMPLE

The program for figure 31 uses a do loop to contour in the X and Z axis. Any contouring in the XZ or YZ axis must be done with a ballnose end mill or an end mill with a corner radius. Let us assume that the part has been roughed out and this program will take a finish cut.

The end mill will feed down along the edge of the part, contour the radius, feed to the center of the other radius, contour, feed up along the edge to the top of the part. The cutter will then increment .050 in the Y axis and the entire sequence in reverse. This entire sequence is repeated 20 times to move a total distance of 2.000 inches; $(2 \times .050) \times 20 = 2.000$.

TOOL CHANGE POSITION



X-2.
Y 2.

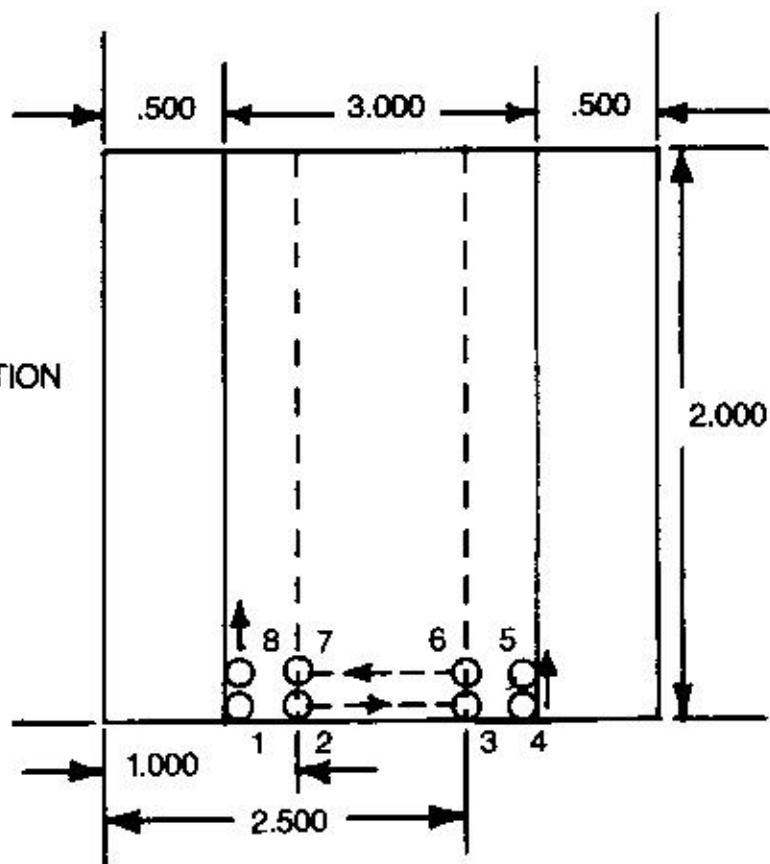


Figure 31: CONTOURING

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE B: CONTOURING PROGRAMMER WCS DATE 6-1-85

TOOL LIST

OFFSETS

TOOL 1: .25 DIA BALL NOSE END
MILL @ 3,500 R.P.M

SETUP INSTRUCTIONS

PART HELD IN VISE.
X, Y ϕ IS LOCATED IN LOWER
LEFT CORNER.

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1	TOOL	1001				21			Z-.125	F	A
2	X.25		Z .70	R	A	22		Y.05		F	I
3	TOOL	ϕ				23	END				
4			Z ϕ	R	A	24	TOOL	ϕ			
5	X-2.	Y2.		R	A	25			Z ϕ	R	A
6	TOOL	1				26	X-2.	Y2.		R	A
7	X.625	Y ϕ		R	A	27	END				
8			Z.1	R	A	28					
9	FEED	20.				29					
10	DO	20				30					
11			Z-.625	F.	A	31					
12	ARC	CW				32					
13	X.375		Z ϕ	F	I	33					
14	X.375		Z-.375	F	I	34					
15	ARC	CW				35					
16	X2.5			F	A	36					
17	ARC	CW				37					
18	X ϕ		Z.375	F	I	38					
19	X.375		Z-.375	F	I	39					
20	ARC	CW				40					



- Event 1-6: Defines the tool offset, retracts the quill, moves to the tool change position and activates tool #1's offset.
- Event 7: Positions the center of the tool inside the left edge of the contour.
- Event 8-11: Feeds the tool to the start point of the first arc and starts the do loop sequence.
- Event 12-15: Contours the first arc. Note that you are programming the center of the ball end mill, not the bottom of the cutter.
- Event 16-20: Feeds the tool to next arc start point and contours it.
- Event 21-23: Feeds the tool to the top of the part, increments .050 in the Y axis, then feeds down to the start point of the next arc.
- Event 24-34: Contours the arc, feeds the tool to the next arc and contours it. The tool then feeds to the top of the part and increments in Y .050.
- Event 35: Ends the do-loop which was started on event 10. In this way everything between events 10 and 35 will be repeated by using a do-loop.
- Event 36-39: Deactivates tool #1's offset, retracts the quill, rapids to the tool change position and ends the program.

SECTION 10

UNDERSTANDING CUTTER DIAMETER COMPENSATION

SECTION 10: UNDERSTANDING CUTTER DIAMETER COMPENSATION

This feature should not be used until all functions of the control, discussed so far, are understood. It is strongly recommended that you use the control at least one week before using cutter diameter compensation.

Previously, you have had to program the tool center. If a different size cutter is going to be used besides the one which was programmed, the program would have to be edited for the new tool size. Now, cutter diameter compensation can be used and the control will calculate the new dimensions for the new tool path. The program does not have to be completely edited.

Cutter diameter compensation can also be used to compensate for the full size of the cutter. By programming the part edge and using cutter diameter compensation, the new tool path at the center of the cutter can be calculated by the control. All of these situations will be covered in the following examples. The default condition for cutter compensation is that it is active in the XY plane. If a ball-nose end mill is used, cutter compensation can be used in the YZ or XZ plane.

All of the following examples show only XY plane operations - but the same rules apply to XZ or YZ plane combinations.

To select the plane you wish to use cutter compensation in, program one of the following codes prior to programming G41 or G42:

- G17 - (default) selects XY plane.
- G18 - selects XZ plane.
- G19 - selects YZ plane.

Each of these three codes are modal and stay active until one of the other codes are programmed.

A. CALCULATING THE TOOL PATH

A new tool path is calculated by the control to the left or right of the original programmed path. In Figures 32-33 you can assume the part edge was programmed. Therefore, the part edge is the original programmed path. In Figure 32, if the part was programmed from point A to point B and continued around the part in that direction, the cutter would have to be placed on the left of the original path to cut around the outside of the part. The command to compensate to the left is G41.

If the part was programmed from point B to point A and continued around the part in that direction (as in Figure 33), the cutter would have to be placed on the right of the original programmed path to cut around the outside of the part. The command to compensate to the right is G42.

G41: CUTTER LEFT

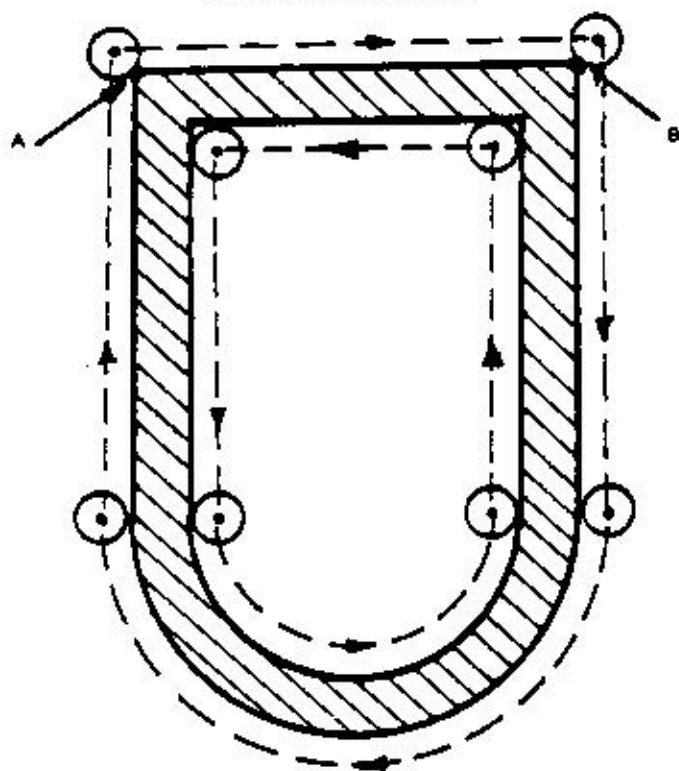


Figure 32

G 42: CUTTER RIGHT

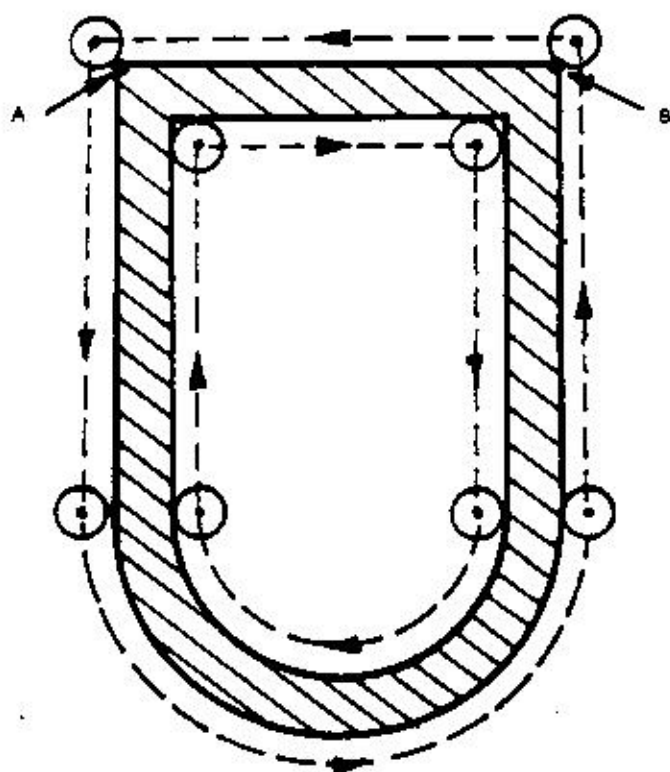


Figure 33

Study figures 34 and 35. If the part edge were programmed, write down the correct command (G 41 or G 42) to move the cutter as shown from position A to B. The answers are below. Also write down the command (G 41 or G 42) to move the cutter as shown from B to A.

Figure 34: From A to B: G 41
From B to A: G 42

Figure 35: From A to B: G 41
From B to A: G 42

B. DISTANCE OF COMPENSATION

The distance the cutter is to stay away from the programmed path will depend on the size of the cutter being used and the way the part is programmed. In Figure 36A, the part edge was programmed and cutter compensation is used for the full diameter of the cutter. The value of compensation in this case is .750 (the full diameter of the cutter).

When programming the part edge, the diameter of the cutter being used will be the value of the offset. In this example, a .750 cutter was programmed.

When the program was run, a .725 cutter was all that was available. Cutter compensation could be used to calculate the new tool path. The offset value will be .025 (the difference between cutter sizes.) When a full size cutter is programmed and an undersize cutter used, the difference in their diameters is the value of the offset. In this example, .025 is the offset and G42 (cutter to the right of the programmed path) is used.

In Example 36C, the tool path was programmed for a .750 diameter cutter. When the program was run, a .875 cutter was chosen. Cutter compensation could be used to calculate the new tool path. The offset value will be .125 (the amount the cutter is oversize.) When a full size cutter is programmed and an oversize cutter is used, the difference in their diameters is the value of the offset. In this example .125 is the offset and G41 (cutter left of the programmed path) is used.

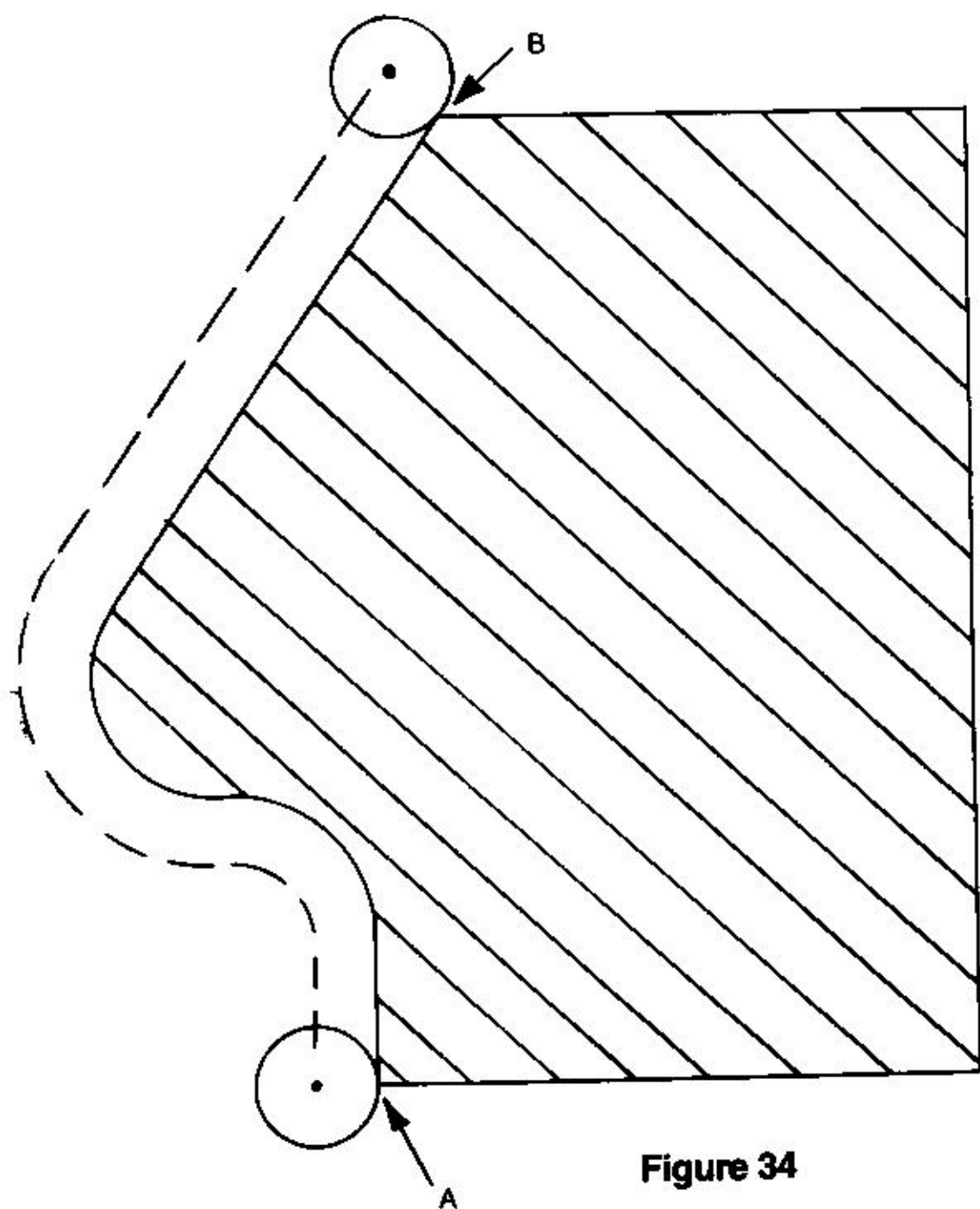


Figure 34

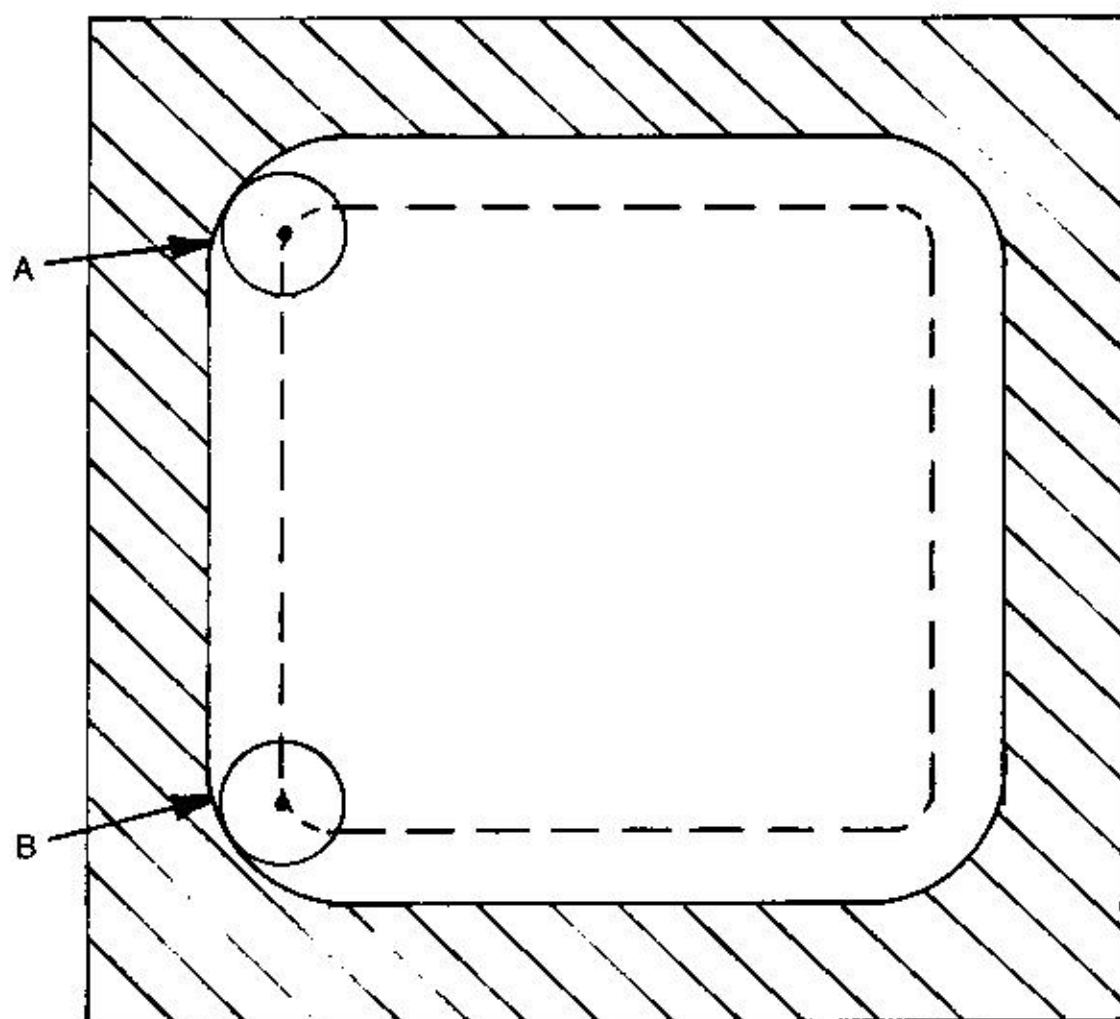
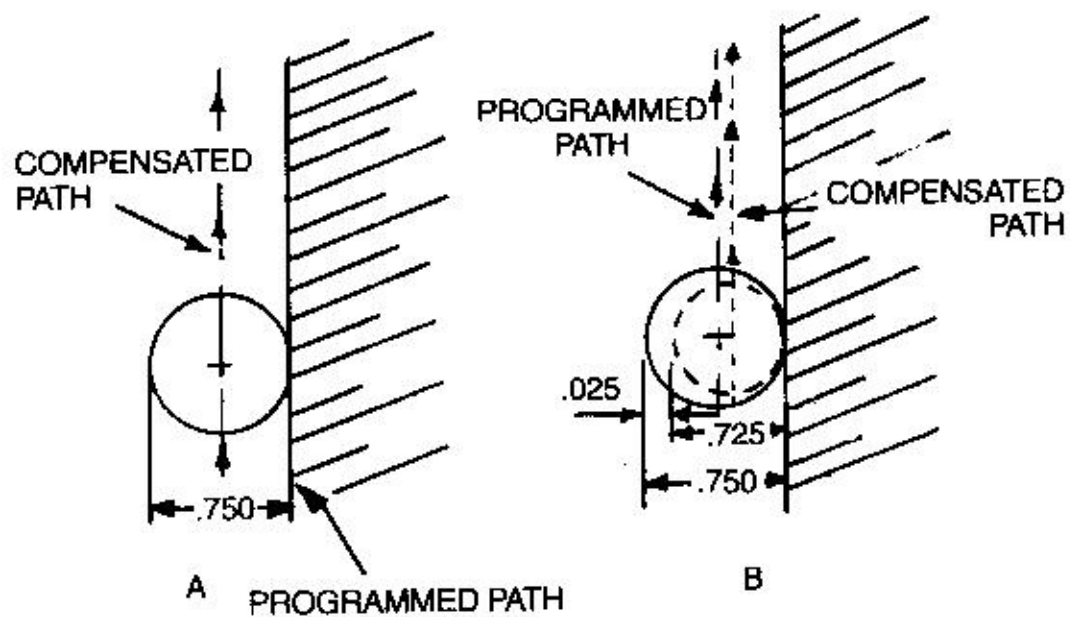


Figure 35

PART EDGED PROGRAMMED UNDERSIZE CUTTER



OVERSIZE CUTTER

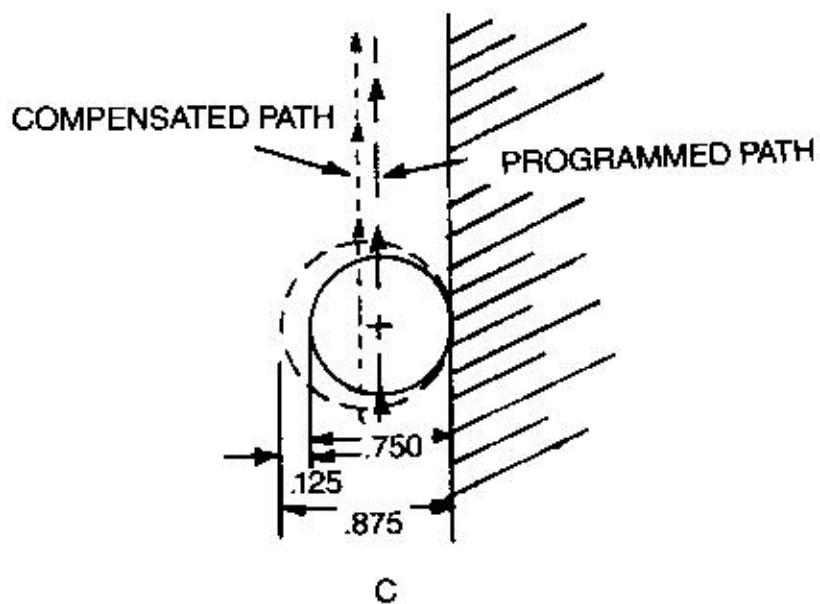


Figure 36

These cutter diameter offset values are usually entered into the program by the operator when the actual tool size is measured during the machine set up, just as the tool length offsets are. The offset value is entered in the same event that the tool length offset value is entered. For a cutter diameter offset value of .750 for tool 1 and a tool length offset of -.919, the events would be:

1. Tool 1001
2. X. 750 Z-.919

The cutter diameter offset value is put in the program as an X value in the event following the tool 1001 (or tool 1002 for tool 2, etc.

C. TURNING COMPENSATION ON AND OFF

The cutter diameter offset value is entered in the program after TOOL 1001. This offset must then be activated later by programming TOOL 1. The TOOL 1 event will tell the control to use the offsets described for TOOL 1.

If no offsets were given in the event following TOOL 1001, the control will assume an offset of zero. After the tool is programmed, the control will make the offsets active. The tool length offset will become active immediately. A G41 or G42 must be programmed so the control knows if compensation is to take place on the left or on the right. The G41 or G42 event activates the cutter diameter offset during the next X and/or Y move. This will be called the "ramp on" move.

In Figure 37, an offset value is given for tool 1 and is activated by programming TOOL 1. The tool is programmed to move to position A and a G41 event is entered. During the next X, Y move to position B, the cutter diameter compensation will take effect.

After a G41 or G42, the center of the cutter will always ramp-on to a point perpendicular to the next X and/or Y move. In Figure 37 this point is B1, which is perpendicular to the next XY Move from B to C. If we had programmed the tool to start at position G and ramp-on to position B, it would still have moved to point B 1. If the cutter was at cutting depth in the workpiece, it would cut the corner off the workpiece as shown. From this example it is evident that understanding the ramp-on move when initiating cutter diameter compensation is critical so that safe ramp-on moves will be programmed.

RAMP ON EVENT

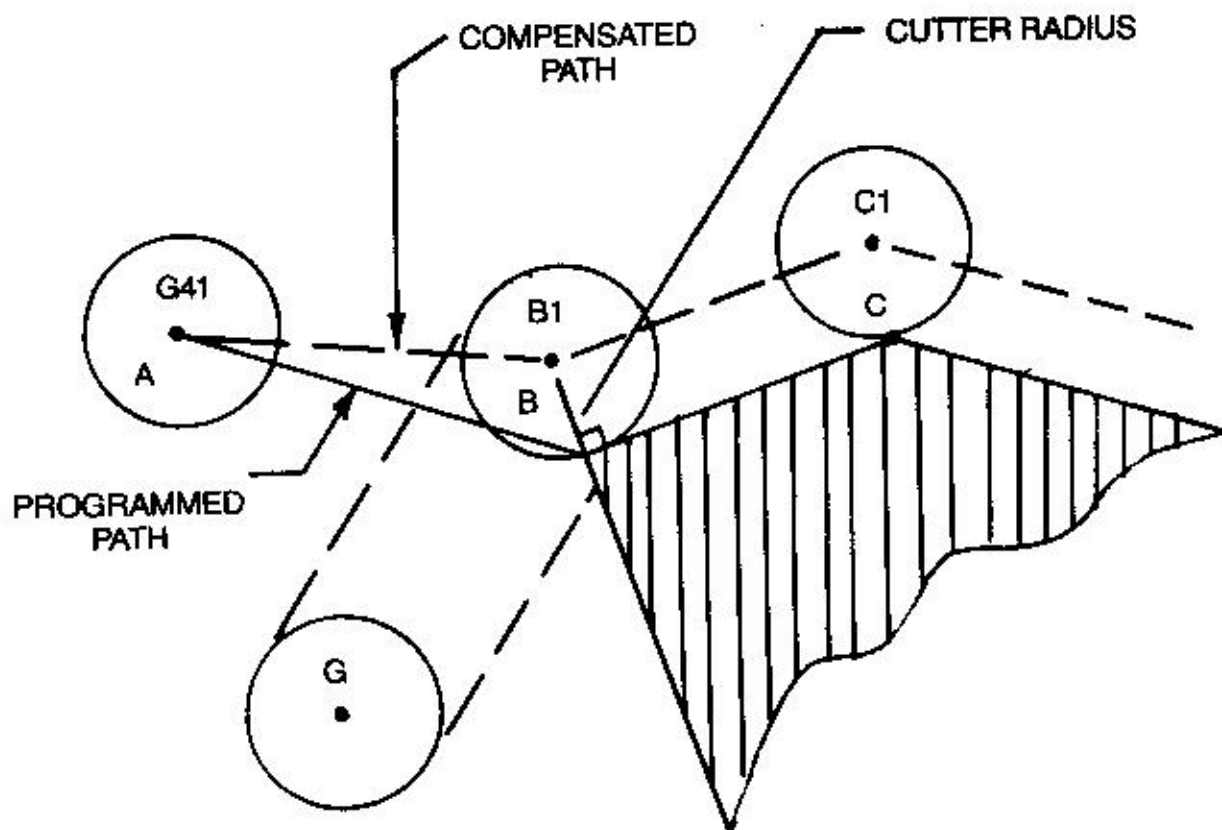


Figure 37

Figure 3A shows what happens when cutter compensation is turned off. G40 is used to turn off compensation. After the event to move the tool to position E was entered, a G40 was entered. A G40 causes the cutter to stop at a position (E 1) which is perpendicular to the previous X and/or Y move. The previous X Y move was from D to E, so the tool will stop at E 1.

After the G40 event, the cutter will move away from the part during the next X and/or Y move. This will be called the ramp-off move. Notice the path the tool would take from point E 1 to point F. If programmed from point E to point F with the cutter at cutting depth, it would cut the corner off the workpiece.

D. COMPENSATING AN INSIDE POCKET

Figure 3B shows the correct way to program an inside pocket. Point B must be at least one cutter radius away from the corner of the pocket. The pocket was programmed: point A, G42, point B, C, D, E, F, B, G40 and back to A. Following the rules above, the ramp-on move is the Y-axis tool motion from A to B. The tool stops at B 1 which is perpendicular to B C. B C is the next X-axis move. (The tool will also stop at B 1 when G40 is programmed after the move from F to B).

Figure 40 shows what would happen if you programmed the ramp-on move from A to C. Following rules, the tool would move to C 1 which is perpendicular to the next Y axis move to C to D. A similar problem would occur if a G40 was programmed after moving from F to C. The tool would stop at C2, perpendicular to the move F to C.

The same part which was drilled in Example 1, will be used to demonstrate cutter diameter compensation. See Figure 41. The part edge will be programmed from A to B, C, D, E, F, G, H. The full diameter of the cutter (.375) will be the offset value and cutter diameter compensation will be turned on between point A and B causing the tool to go from point A to B1, C1, D1, E1, F1, G1. Compensation will be turned off between G and H which will cause the tool to move to point H. Use the part drilled in Example 1 and cut this shape into the part using cutter diameter compensation and an available 2 flute end mill. Enter its diameter and length offsets in Event 2.



●

C.

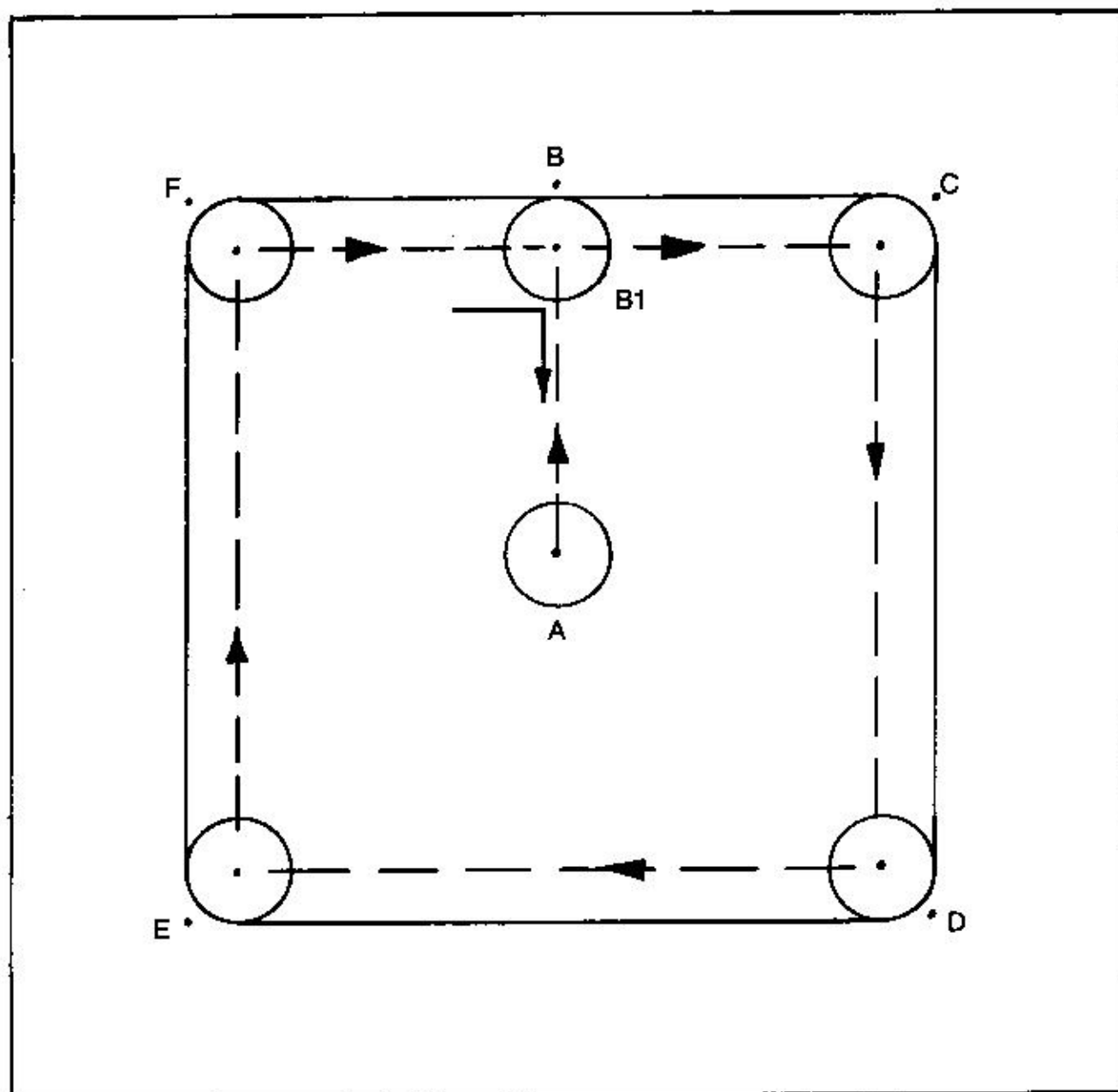


Figure 39

D.

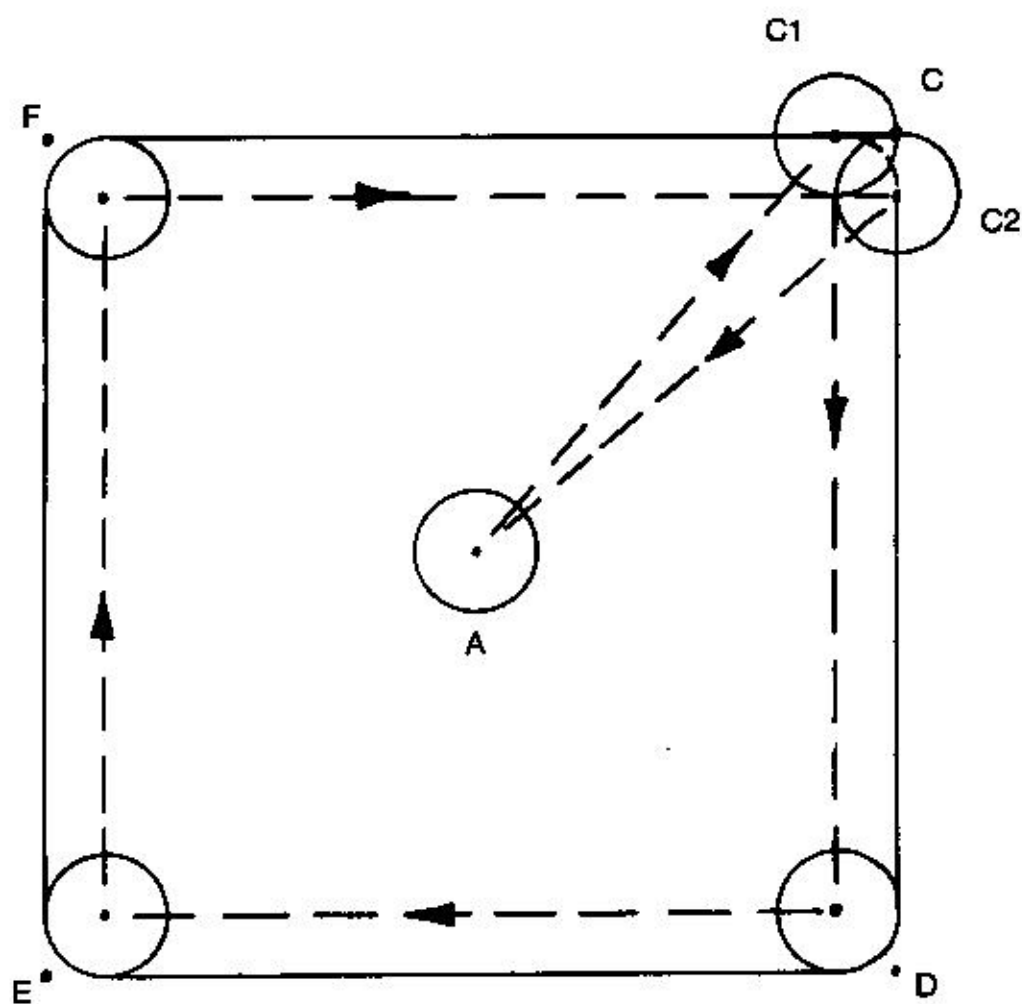


Figure 40

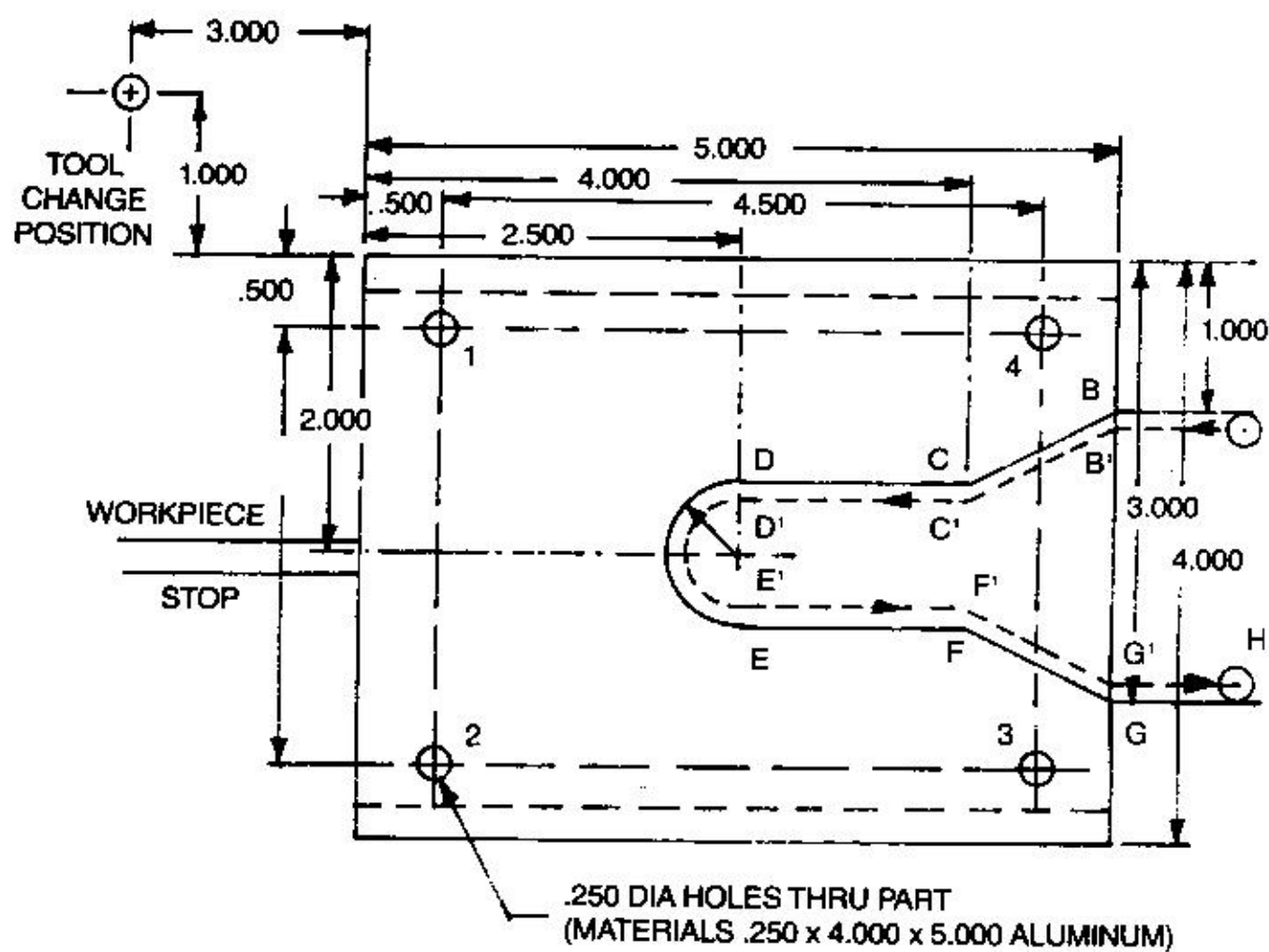


Figure 41

Event 1: Tool 1001 is tool 1's offset definition.

Event 2: Left open for operator to enter offset values.

Event 3: Deactivates offsets.

Event 4: Retracts quill.

Event 5: Rapids to tool change position.

Event 6: Activates tool 1's offsets.

Event 7: Rapids tool to position A.

Event 8: Rapids tool to cutting depth.

Event 9: Activates cutter compensation to the left.

Event 10: Sets the milling feedrate.

Event 11: Causes the tool to move to point B1 because G41 is active. This is the ramp on event.

Event 12: Tool feeds to point C1.

Event 13: Tool feeds to point D1.

Event 14: Adjusts feedrate for constant surface speed.

Event 15: Starts arc counter clockwise.

Event 16: Defines arc center.

Event 17: Defines arc finish point.

Event 18: Arc end command.

Event 19: Sets feedrate back to 4 IPM.

Event 20: Tool feeds to point F1.

Event 21: Feeds tool to point G1 because a G40 is programmed before the next X Y move.

Event 22: Deactivates cutter diameter compensation.

Event 23: Tool feeds to point H. This is the ramp-off event.

ANILAM CRUSADER PROGRAM SHEET

 PART OR DRWG. # EXAMPLE 9: W/OUTTRA COMP PROGRAMMER QST DATE 4-18-85

TOOL LIST						OFFSETS						SETUP INSTRUCTIONS					
TOOL 1: .375 END MILL @ 2300 RPM												X, Y ϕ AT UPPER LEFT CORNER ACTUAL MEASURED DIAMETER OF TOOL USED ENTERED AS "X" VALUE AS IF FEED 2 WITH Z-AXIS TOOL LENGTH OFFSET					
EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS						
1	TOOL	1001				21	X 5.	Y - 3.		F	A						
2	X .375		Z .760	R	A	22	G 40										
3	TOOL	ϕ				23	X 6.	Y - 2.8125		F	A						
4			Z ϕ	R	A	24	TOOL	ϕ									
5	X - 3.	Y 1.		R	A	25			Z ϕ	R	A						
6	TOOL	1				26	X - 3.	Y 1.		R	A						
7	X 6.	Y - 1.1875		R	A	27	END.										
8			Z - .35	R	A	28											
9	G 41					29											
10	FEED	4.				30											
11	X 5.	Y - 1.		F	A	31											
12	X 4.	Y - 1.5		F	A	32											
13	X 2.5			F	A	33											
14	FEED	2.5				34											
15	ARC	CCW				35											
16	X 2.5	Y - 2.		F	A	36											
17	X 2.5	Y - 2.5		F	A	37											
18	ARC	CCW				38											
19	FEED	4.				39											
20	X 4.			F	A	40											



Event 24: Deactivates tool offsets.

Event 25: Retracts quill.

Event 26: Rapid to tool change position.

Event 27: End of program.

E. CUTTER COMPENSATION PROGRAMMING EXAMPLE

Figure 42 shows a hex-shaped part which will be programmed using cutter compensation. The part edge will be programmed and the full diameter of the tool will be entered as the offset value. The part will be held in a fixture and absolute zero will be set in the center of the part. The speeds and feeds will be set for a .375 diameter cutter but actual cutter diameters can be entered and used with cutter diameter compensation. The actual cutter used should be as close to .375 as possible to use the programmed feeds and speeds.

The tool change position will be at X-2.000 and Y 2.000 absolute. The tool will be programmed to move to points A,B,C,D,E,F and back to A using cutter compensation. G41 will be turned on at point 1. As the previous rules in section 3 state, the cutter will ramp-on during the next X Y move to point A and stop at point 2 which is perpendicular to the next X Y move from A to B. The tool will stay at the compensated distance away from the part around points 3,4,5,6,7 and 8. The program will be the dimensions of points A through F and back to A.

At point A, G40 is programmed. This will cause the cutter to stop at point 8. Again, this follows the rules of turning compensation off. Point 8 is perpendicular to the previous X Y move from F to A. The tool ramps-off after G40 during the next X Y move from 8 to 1.

Event	1:	TOOL 1001 defines tool 1's offsets.
Event	2:	Left open for entering the cutter diameter offset value in X and the tool length offset in Z.
Event	3-6:	Retracts the quill, moves to the tool change position and puts Tool 1's offsets in effect.
Event	7:	Moves the tool -.400 below the top of the part.
Event	8:	Sets compensation to the left.
Event	9:	Sets the feedrate.

ANILAM CRUSADER PROGRAM SHEET

 PART OR DRWG. # EXAMPLE 10: COTTER POND PROGRAMMER RPS DATE 4-1-85

TOOL LIST						OFFSETS						SETUP INSTRUCTIONS					
TOOL 1: .375 END MILL @ 1.500 RPM						PART HELD IN FUTURE						X, Y ϕ LOCATED AT CENTER OF PART					
EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS					RAPID / FEED	INCR / ABS
1	TOOL	1001				21	END										
2	X.375		Z TLO	R	A	22											
3	TOOL	ϕ				23											
4			Z ϕ	ϕ	A	24											
5	X-2.	Y 2.		R	A	25											
6	TOOL	1				26											
7			Z-.4	R	A	27											
8	G41					28											
9	FEED	5.1				29											
10	X-.5774	Y 1.		F	A	30											
11	X.5774			F	A	1											
12	X 1.1547	Y ϕ		F	A	2											
13	X.5774	Y-1.		F	A	3											
14	X-.5774			F	A	4											
15	X-1.1547	Y 0		F	A	5											
16	X-.5774	Y 1.		F	A	6											
17	G40					7											
18	X-2.	Y 2.		R	A	8											
19	TOOL	ϕ				9											
20			Z ϕ	R	A	0											



EXAMPLE 9

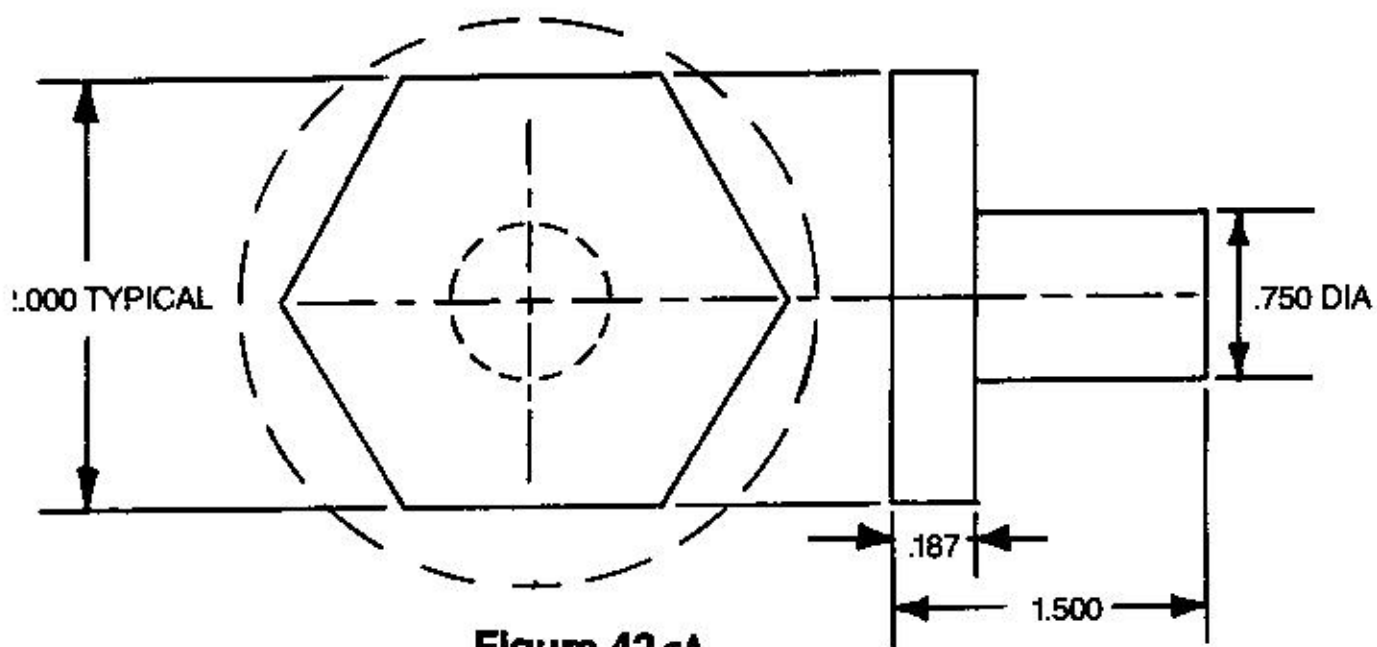
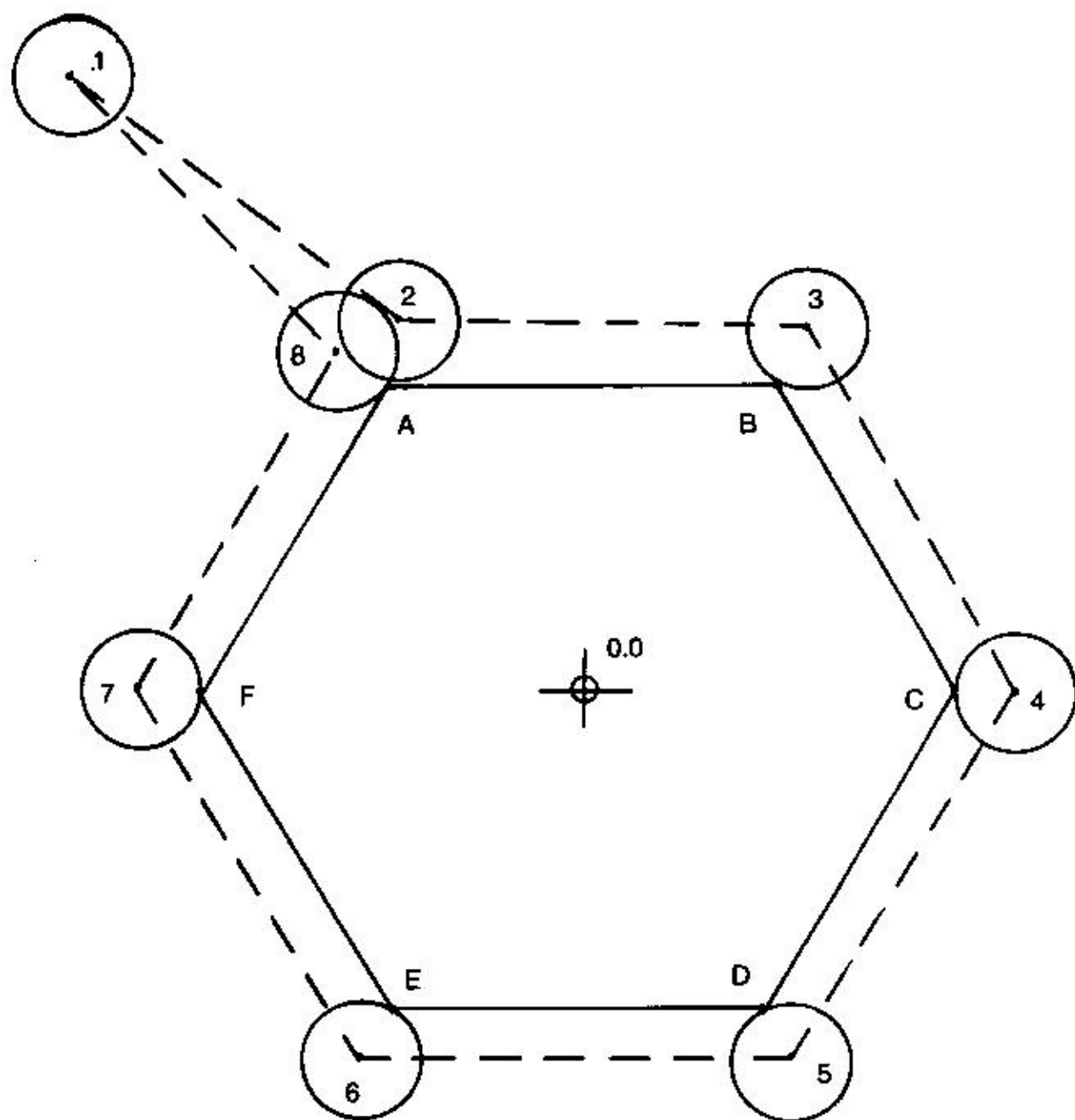


Figure 42 -A



42-B

Event	10:	The ramp-on move in which the tool will move to position 2.
Event	11-16:	The remaining coordinates calculated for points B through A.
Event	17:	Deactivates cutter diameter compensation (not tool length compensation.) The tool will stop at point B.
Event	18:	The ramp off move back to X-2. Y2.
Event	19-21:	Retracts the quill and ends the program.

Examine the illustrations to see how the tool path is adjusted when using G41 and G42. The solid lines represent the programmed path, the dashed lines represents the tool path.

F. RULES OF CUTTER DIAMETER COMPENSATION

These rules apply only when using cutter diameter compensation:

1. The offset value is given as an X axis dimension after the TOOL 1001 (or TOOL 1002 etc.) definition event.
2. Tool 1 (or Tool 2, etc.) must be programmed before G41 or G42 so the control will know which tool number's offset is active.
3. The cutter ramps-on during the next X and/or Y move after a G41 or G42.
4. The cutter ramps-on perpendicular to the next X and/or Y move.
5. There must be at least one straight line move after G41 or G42 before cutting an arc.
6. An arc can only be followed by a straight line move or another arc. G40 is not permitted immediately following an arc.
7. Z-axis moves are permitted at any time.
8. Changing tool numbers (including changing to Tool 0 or ending the program) is not permitted until after the ramp-off move following G40.
9. G40 causes the tool to stop perpendicular to the last X and/or Y move.
10. Each compensated move must have an intersection.

G. POCKET CUTTER COMPENSATION PROGRAMMING EXAMPLE

The part in figure 43 has a window which is roughly cast. A rough cut and finish cut is required to finish the part. A .250 diameter end mill will be used. The tool path at the center of the cutter will be programmed in a sub-routine. The cutter will be defined as having a .020 cutter diameter offset for the rough cut. This will leave .010 on a side for the finish cut. With .020 offset active, the sub-routine will be called to rough out the window. The offset value will be changed to zero and the same routine will be called to finish the window. In this way the same tool and subroutine can be used with different amounts of compensation to rough and finish a part.

To illustrate this, the tool path is plotted out in the bottom drawing in figure 46. The tool path in subroutine 1 will be points 2,3,4,5,6,7,8,9 back to 3, then 2.

A G41 will be programmed between 2 and 3 causing the cutter to ramp-on and stop at point 3A for the rough cut. Compensation will take place all the way around the part. After point 3A is reached after cutting the window, a G40 will be entered and the tool will ramp-off back to point 2.

Compensation can be used because the cutter does not ramp-off immediately following an arc.

Event 1: Tool 1001 to define tool 1's offsets.

Event 2: X .020 sets the offset to rough out the part with .010 material left for a finish cut. The tool length offset will be entered during set up.

Event 3-6: Retracts the quill, moves to the tool change position and activates tool 1's offsets.

Event 7: Rapids the tool to position 2.

Event 8: Rapids the tool to cutting depth. The window is cast in the part and the tool rapids through the window.

Event 9: Sets the feedrate for roughing at 10.01 IPM.

Event 10: Calls subroutine 1 which roughs the window with .020 cutter diameter compensation. The subroutine initiates G 41, ramps on, contours around the window, cancels using G 40 and ramps off, back to point 2.

Event 11: Redefines tool 1's offsets.

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 11: BOXING CUTTING COMP PROGRAMMER RC H DATE 11-25-84

TOOL LIST

OFFSETS

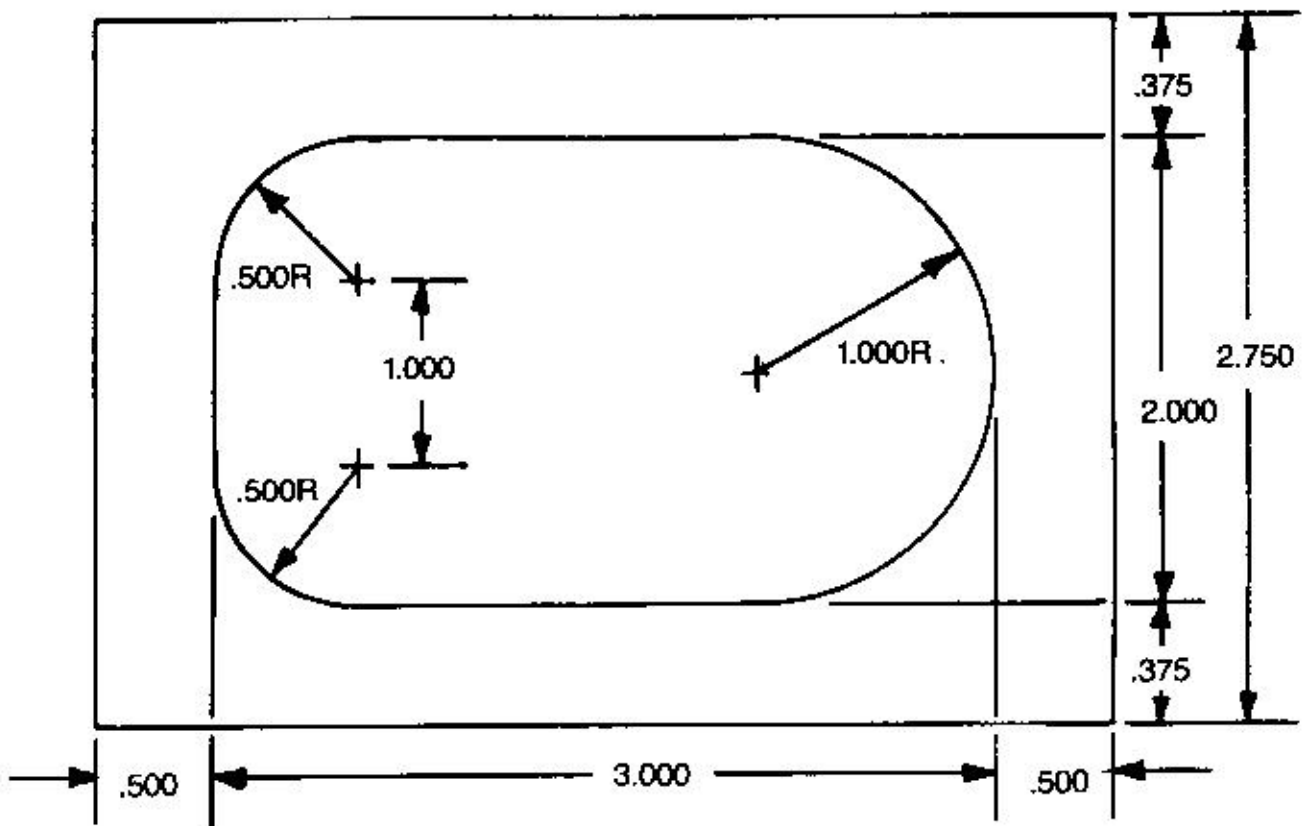
SETUP INSTRUCTIONS

Tool 1: .25 END MILL @ 3000 RPM

PART HELD IN VISE
X, Y ϕ LOCATED IN LOWER LEFT
CORNER

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1	Tool	1001				21	G41				
2	X .020		Z TLO	R	A	22		Y 2.25		F	A
3	Tool	ϕ				23	X 1.			F	A
4			Z ϕ	R	A	24	ARC	CCW			
5	X-2.	Y 1.		R	A	25	X 0	Y-.375		F	I
6	Tool	1				26	X-.375	Y-.375		F	I
7	Y 2.	Y 1.75		R	A	27	ARC	CCW			
8			Z-.3	R	A	28		Y-1.		F	I
9	FEED	10.				29	ARC	CCW			
10	CALL	1				30	X.375	Y 0		F	I
11	Tool	1001				31	X.375	Y-.375		F	I
12	X 0		Z TLO	R	A	32	ARC	CCW			
13	Tool	1				33	X 1.5			F	I
14	FEED	20.				34	ARC	CCW			
15	CALL	1				35	X 0	Y.875		F	I
16	Tool	ϕ				36	X 0	Y 1.75		F	I
17			Z ϕ	R	A	37	ARC	CCW			
18	X-2.	Y 1.		R	A	38	X 2.			F	A
19	END					39	G40				
20	SUB	1				40		Y 1.75		F	A
						41	END				

A



MATERIAL: .250 THICK

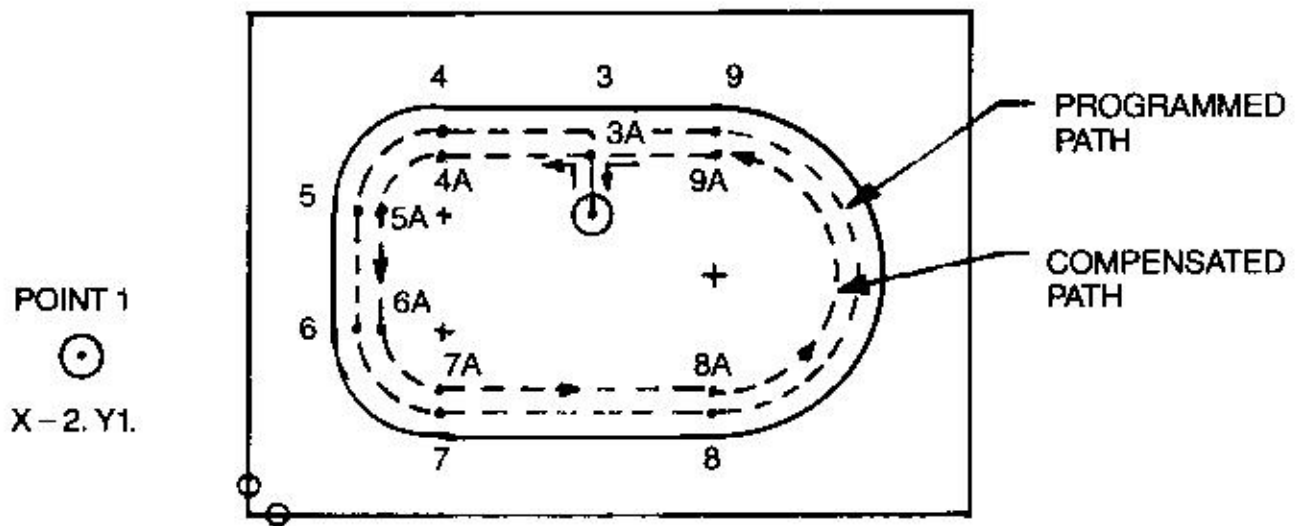


Figure 43: POCKET CUTTER COMPENSATION

Event 12: Sets the tool diameter compensation to zero to finish the window. The tool length offset must be reentered during set up in this event also.

Event 13: Activates the new offset for tool 1. Because the same tool number is programmed twice consecutively, the program will not stop.

Event 14: Sets the feedrate to 20 IPM for the finished cut.

Event 15: Executes the subroutine with zero diameter offset for a finish cut.

Event 16-19: Retracts the quill and ends the program.

Event 20: Starts subroutine with zero diameter offset for a finish cut.

Event 21: Activates G41, cutter left.

Event 22: Ramp on move to point 3 or 3A depending on which offset is active. With zero offset active, the cutter will move to point 3. With .020 offset active, the cutter will move to point 3A.

Event 23-38: Moves the cutter around the window in the part. The offset which is active will determine which points the tool moves to.

Event 39: Deactivates G41.

This causes the tool to stop perpendicular to the move from 9 to 3 and stop at 3A. With zero offset in effect the tool would stop at point 3.

Event 40: The ramp off move to point 2.

Event 41: Ends the subroutine definition.

SECTION 11

UNDERSTANDING G-CODES
AND CANNED CYCLES

SECTION 11: G CODES AND CANNED CYCLES

A. DEFINING G CODES

G-Codes are programs that are stored in the control's permanent memory and are generally referred to as canned cycles. G-codes are designed to make programming easier.

Before we enter a G-code we first have to enter variables. These variables assign values that make the machine produce the part shape and size required.

Each-G code will be explained in this section in detail. The Help mode can be used for all G-code programming. How to use Help will be explained in a later section of this manual. Please note that all whole numbers must be entered with the decimal point in the correct place.

It is not mandatory to use the Help mode in order to load a canned cycle into the program. When not using Help press V, V code number (ie., V24), value required and ENTER.

All V codes must be entered first, followed by the appropriate G code number.

G CODE LISTING FOR SERIES - M

G05	Ellipse
G06	Spiral (Helical or Planar)
G10	Rapid polar move from current position
G11	Feed polar move from current position
G12	Rapid polar move with center specified
G13	Feed polar move with center specified
G17	Select plane XY (default)
G18	Select plane
G19	Select plane YZ
G40	Cutter compensation OFF
G41	Cutter compensation ON left
G42	Cutter compensation ON right
G45	Mold rotation X
G46	Mold rotation Y
G47	Mold rotation Z
G48	Conical cavity milling (toroid)
G49	Elbow cavity milling
G51	Polar rotation activate
G52	Polar rotation deactivate
G53	Scaling activate
G54	Scaling deactivates
G75	Frame milling
G76	Hole milling
G77	Circular pocket milling
G78	Rectangular pocket milling
G79	Bolt hole circle drilling
G80	Deactivate drilling cycle
G81	Basic drilling cycle
G82	Counter boring cycle
G83	Peck drilling cycle

G83 Peck drilling cycle
 G85 Boring cycle
 G86 Boring one direction cycle
 G87 Chip breaking cycle
 G89 Flat bottom boring cycle

B. G05 (Ellipse)

Before programming an ellipse (fig. 44), the cutter must be at the required starting point in all axis (X, Y and Z). All dimensions are incremental. Cutter compensation cannot be used with this canned cycle, because it has its own built-in cutter compensation. The following variables must be programmed before G05.

NOTE :

When cutting an ellipse, the control calculates dimensions to the center of the cutter which is a true ellipse. The length and width will be produced as programmed, but arcs will be larger on external and smaller on internal arcs. This must be taken into account when cutting an ellipse. The reason is a line drawn parallel to an ellipse is not an ellipse.

V01: Direction of cutter movement. A counterclockwise move is determined by a + sign. Clockwise movement is a - sign. If no sign is entered, the control will assume a plus sign (CCW). Plane selection: XY = 1, XZ = 2 and YZ = 3. EXAMPLE: -1. equals CWXY.

V02: Incremental distance from starting point on ellipse to center of ellipse on X axis.

V03: Incremental distance from starting point on ellipse to center of ellipse on Y axis.

V04: Incremental distance from starting point to finish point X.

V05: Incremental distance from start to finish point Y.

V06: Length X

V07: Width Y

V49: Is used inside, outside or on line of programmed ellipse (+1. equals Outside; -1. equals Inside; and 0 equals On Line.

The following program will cut above ellipse:

V01 -1 CW, XY axis
 V02 1. Start to center X axis
 V03 0 Start to center Y axis
 V04 0 Start to end point X axis

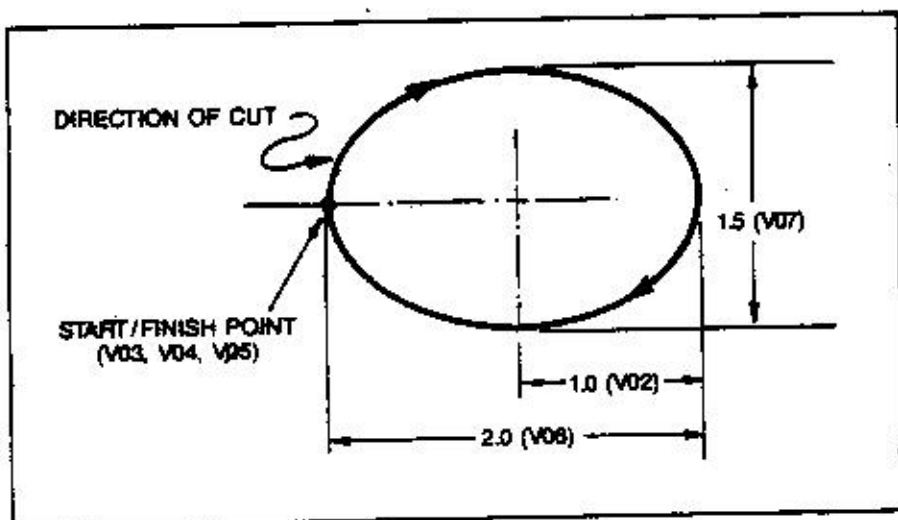


Figure 44

V05	0	Start to end point Y axis
V06	2.	Length X
V07	1.5	Width Y
V49	0	Cutting - on line
G05		Active Ellipse Cycle

C. G06 SPIRALS (PLANER OR HELICAL)

There are two types of spirals: planer (2 axis) and helical (3 axis). The cutter must be located at X, Y and Z positions before programming this canned cycle. All dimensions are incremental in this canned cycle. Cutter compensation cannot be used. See figure 45.

The following variables must be programmed.

V01:	Direction and plane in which spiral is to be cut.
V02:	Incremental distance from start to X axis center.
V03:	Incremental distance from start to Y axis center.
V04:	Incremental distance from start to X axis finish.
V05:	Incremental distance from start to Y axis finish.
V06:	Incremental distance from start Z point to Z finish point. When cutting a planer spiral this would be 0.
V07:	Number of times the cutter crosses the line drawn through the center and starting point. Stopping on the line is not counted as crossing.
G06:	Activate canned cycle.

X	.25	Y	0	Starting point.
Z	-.1.			Z Depth
V01	-1			Direction and plane.
V02	-.25			From start to center X (Incremental)
V03	0			From start to center Y (Incremental)
V04	.375			From start to finish X (Incremental)
V05	0			From start to finish Y (Incremental)
V06	0			From start to finish Z (Incremental)
V07	1.			Number of times crossed starting Y axis line.
10.	G06			Activate canned cycle.

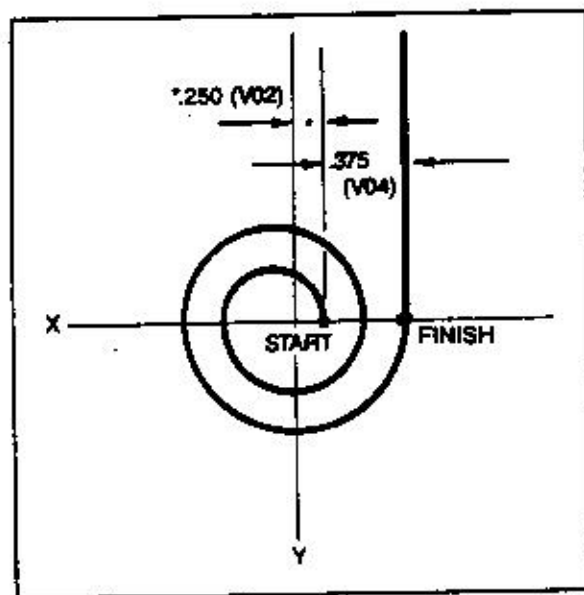


Figure 45

D. G10: RAPID POLAR MOVE WITH SPINDLE AT POLAR CENTER.

A polar move is used in order to save having to calculate coordinates. Given an angle and a radius a position can be located using G10.

NOTE:

All angles must be measured from the 3 o'clock position in relation to the polar center; a clockwise move from the 3 o'clock position is entered as a minus dimension. A counter clockwise move is a plus dimension.

This canned cycle is used when the tool is positioned at the polar center (point A in figure 47 and 48) V14 is the radius dimension and V15 is the angle from the 3 o'clock.

In figure 47, a rapid move from X1. Y1. to a point 30 degrees and 1.125 would be programmed in the following order:

V14	1.125	Radius
V15	30	Angle
G10		Activate radius polar move.

Figure 48, a rapid move from X2. Y2. to a point 135 and .875 would be programmed as follows:

V14	.875	Radius
V15	-135.	Angle
G10		Activate

Note that the angle is entered as a negative dimension because it is below the 3 o'clock position in the figure and is therefore rotating clockwise.

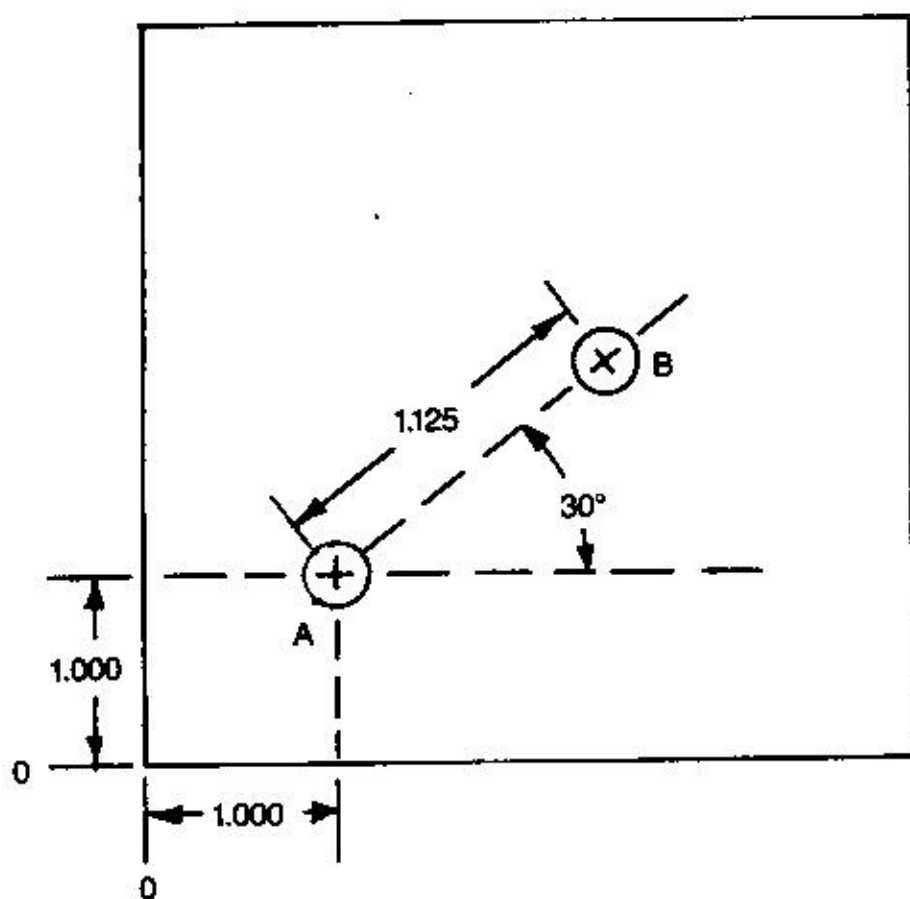
E. G11: FEED POLAR MOVE WITH SPINDLE AT POLAR CENTER.

G11 works similar to G10. The variables are all entered in the same order except the tool motion is made in FEED at current feedrate. See the order of programmed steps below (see figure 49):

FEED	15.	Feedrate
V14	2.3	Radius
V15	-63.5	Angle
G11		Activate feed polar move.

F. G12: RAPID POLAR MOVE WITH CENTER SPECIFIED

This G code works when you need to make a polar move but the present tool location is not at the polar center. These V codes must be entered prior to programming G12. Review the program



G10
RAPID POLAR
MOVE WITH
SPINDLE AT
POLAR CENTER

Figure 47

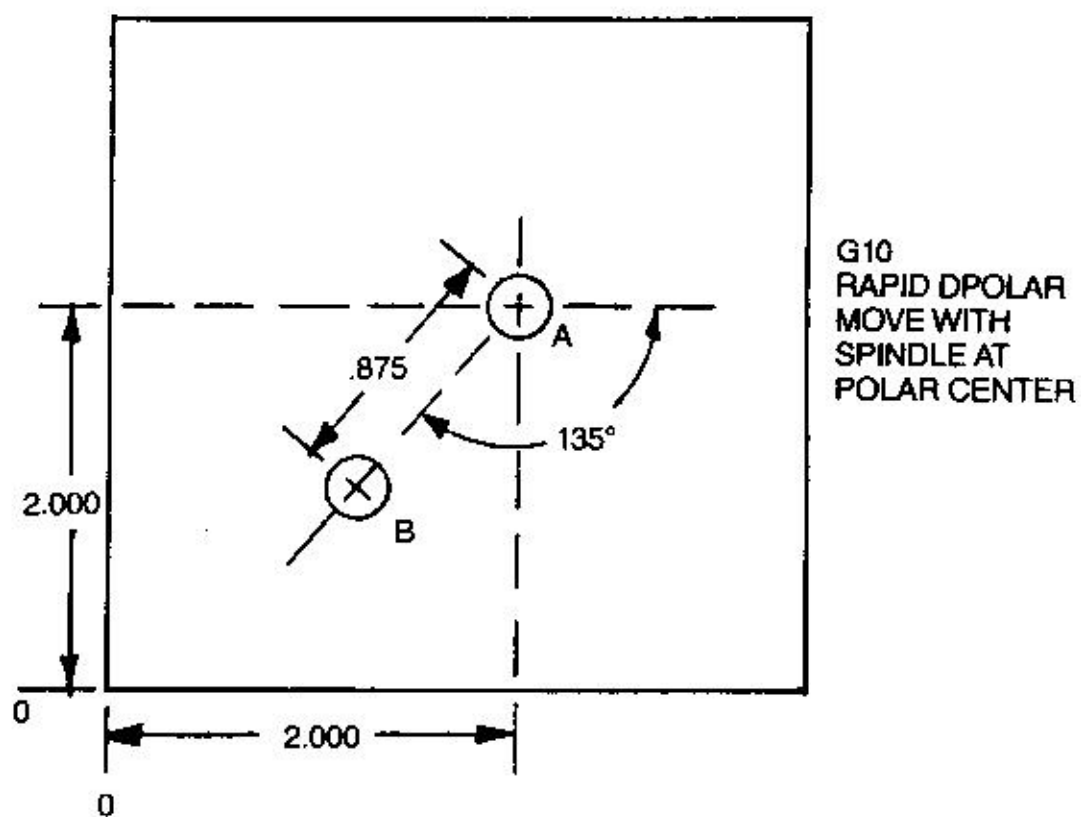
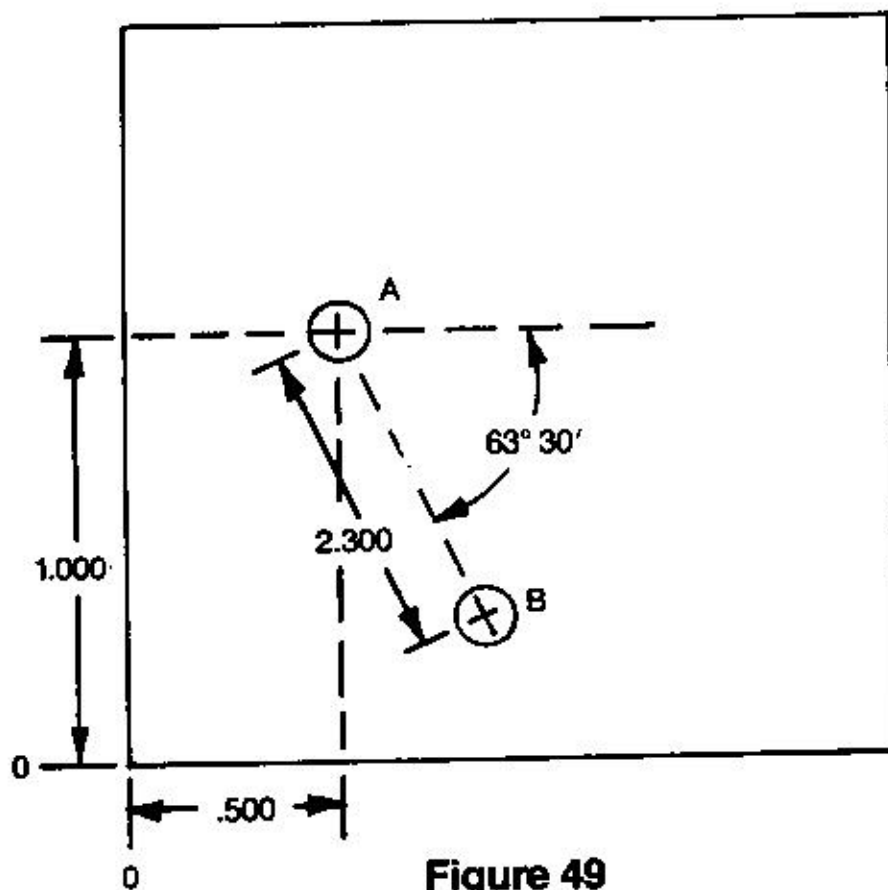


Figure 48



G11
FEED POLAR MOVE
WITH SPINDLE AT
POLAR CENTER

Figure 49

steps below. This will show how to move from X3. Y1. to a point at 135 degrees from polar center (point A to B). See figure 50.

V11	1.	Polar center X
V12	1.5	Polar center Y
V13	0	Index angle
V14	.6	Radius
V15	135.	Angle from 0
G12		Rapid move activated.

6. G13: FEED POLAR MOVE WITH CENTER SPECIFIED

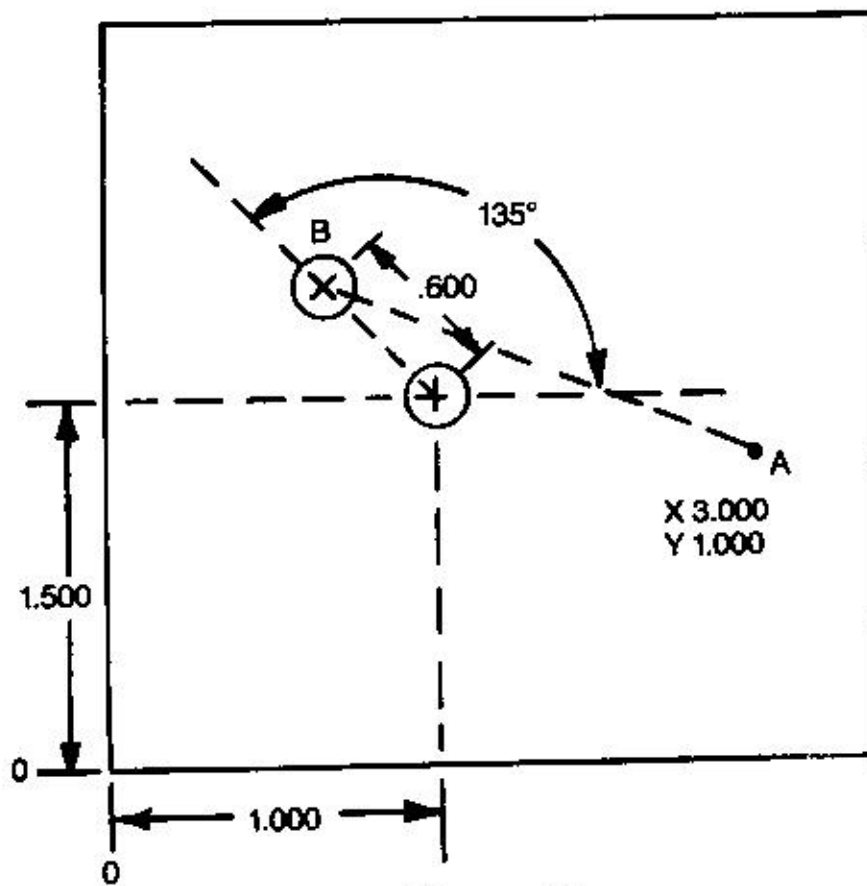
G13 works similar to G12, except the move is made in Feed at the current feedrate. The move from X1. Y0 to the point 15 15' from Index Angle of the 90 to polar center (point A to B) is programmed as follows (see figure 51):

FEED 20	Feedrate
V11 2.	Center X
V12 1.	Center Y
V13 90.	Index Angle
V14 .4375	Radius
V15 -15.25	Angle
G13	Feed move activated

You may also use G13 so that the end point of an arc can be reached with a polar move. The format for making the arc is the same as it would be if you knew the XY coordinates of the end point. However, instead of programming the XY end point, the polar command G13 is used. See figure 52 and refer to the program steps below to see this capability in use:

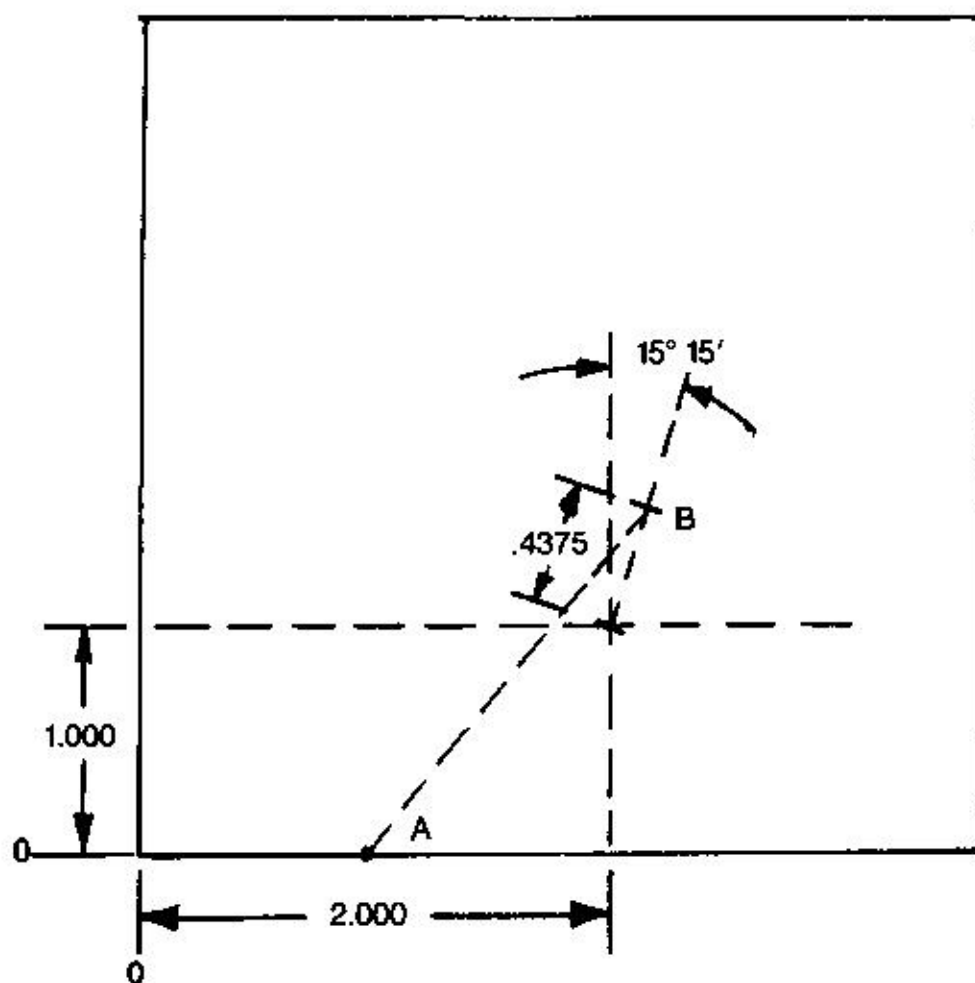
FEED 5.	Feedrate
X 1.625 Y1.FA	Feed to start of arc
V11 1.	Center X
V12 1.	Center Y
V13 0.	Angle measured from 0 degrees
V14 .625	Radius
V15 36.	Angle
ARC CCW	Arc Command CCW
X 1. Y 1. FA	Arc Center
G13	Polar move to end point
ARC	Arc Finish Command

Because the Arc CCW command is active, the tool will move in a circular rather than linear motion.



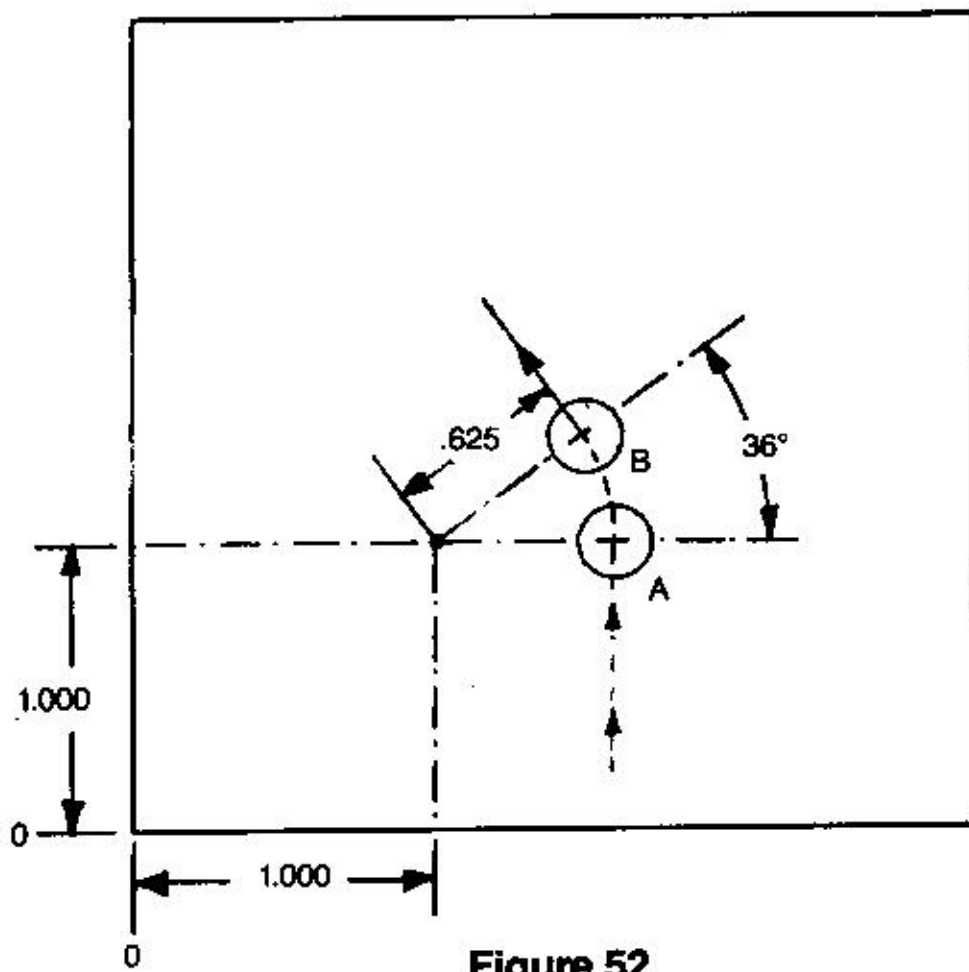
G12
RAPID POLAR
MOVE WITH
CENTER SPECIFIED

Figure 50



G13
FEED POLAR MOVE
WITH CENTER
SPECIFIED

Figure 51



6. G CODES RELATED TO CUTTER COMPENSATION

The G17, G18 and G19 codes are to select the plane in which cutter compensation is to be used.

G17: XY plane (default)
G18: XZ plane
G19: YZ plane

G40: CUTTER COMPENSATION OFF

G41 and G42: CUTTER DIAMETER COMPENSATION

Cutter compensation has been discussed in detail in section 10 of this manual.

1. G45, G46 AND G47: MOLD ROTATION

These canned cycles allow us to take a profile and rotate it around an axis.

G45: Rotates around X axis.
G46: Rotates around Y axis.
G47: Rotates around Z axis.

Again, specific variables have to be entered. Zero angle is set from where the program is written. Angles given as a plus are counter clockwise, angles give as a minus are clockwise. Two subroutines have to be written. One will cut the profile in one direction and the other will cut it in the opposite direction. All dimensions must be incremental. The number of complete cycles is equal to a subroutine in each direction.

Figure 53 shows a cavity that has to be produced. Figure 54 shows the cutter path that has to be programmed in order to cut the cavity using a 3/8" dia ball end mill. There is tooling information required to run this program which will be covered later.

X 0 Y 0 RA	Moves tool to start position point 1.
Z .25 RA	Moves tool to .0625 above part.
Z 0 FA	Moves tool down into part.
V61 0	START ANGLE. Since tool is at the starting position, the angle is 0.
V62 -180.	FINISH ANGLE. In this case, the rotation required is clockwise, so use a minus angle.
V63 9.	NUMBER OF COMPLETE CYCLES. The tool will make 18 passes across the part: 9 in each direction.

Figure A

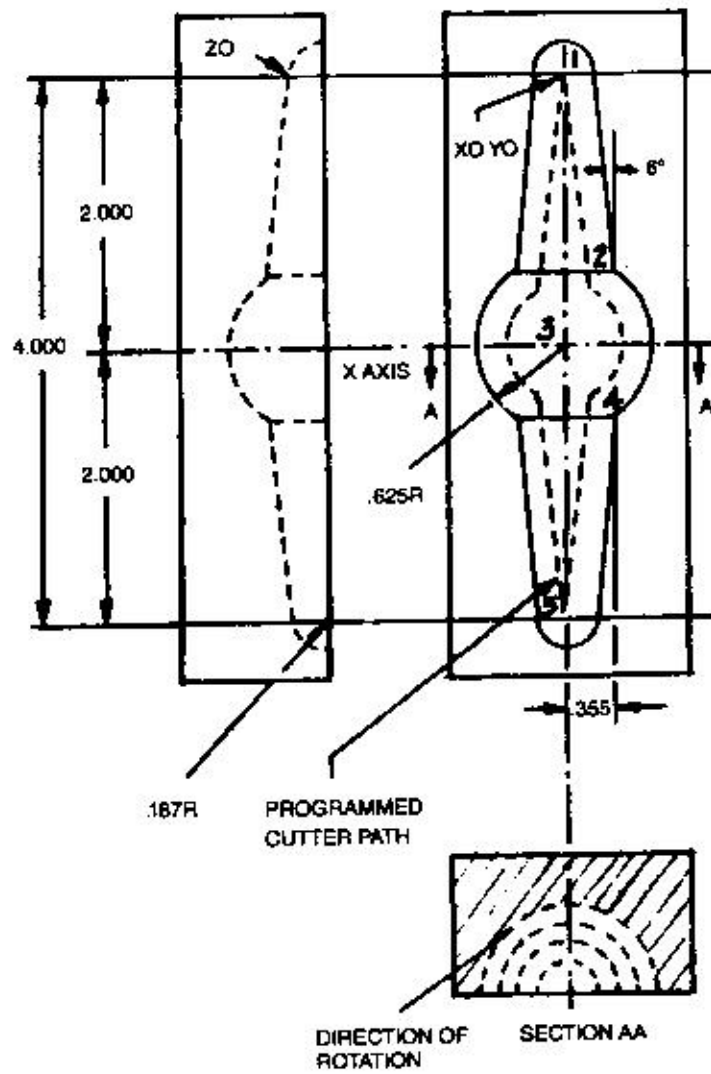


Figure 53

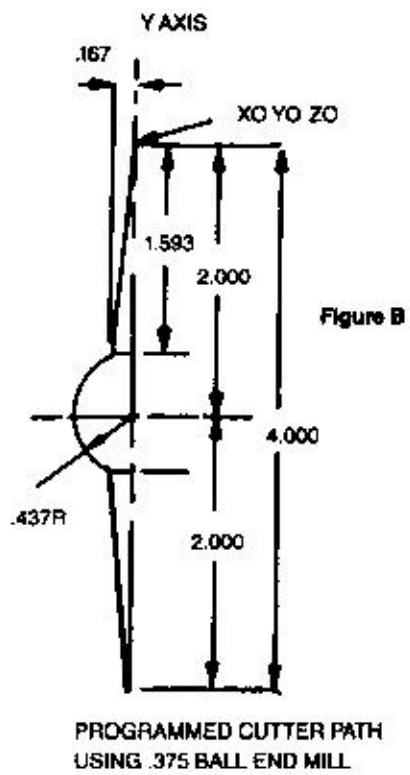


Figure 54

V70	10.		FORW SUBR. This is the subroutine number in forward direction.
V71	11.		REV SUBR. This is the subroutine in reverse direction.
G46			Activate rotation (Y-axis).
END			
SUBR 10			Number of Forward Subroutines.
X-.167 Y-1.593	FI		Move from point 1 to point 2.
ARC CCW			Counter-clockwise arc statement.
X .167 Y-.407	FI		Incremental distance from point 2 (start of arc) to point 3 (finish of arc).
X 0 Y-.814	FI		Incremental distance from point 2 (start of arc) to point 4 (finish of arc).
ARC			Arc end statement.
X .167 Y -1.593	FI		Incremental distance from point 4 to point 5.
AUX 9030			Turns off rotation cycle.
X 0 Y -4. Z 0	FA		Reposition move.
END			End of subroutine.
SUB 11			
X-.167 Y 1.593	FI		Incremental distance from point 5 to point 4.
ARC CW			Arc start statement.
X .167 Y .407	FI		Incremental distance from point 4 to point 3 (center of arc).
X 0 Y .814	FI		Incremental distance from point 4 (start of arc) to point 2 (finish of arc).
ARC			End arc statement.
X .167 Y 1.593	FI		Incremental distance from point 2 to point 1.
AUX 9030			Turn off rotation.

X 0 Y 0 Z 0 FA Reposition move.

END End of subroutine.

More examples of Mold Rotation are at the back of this manual.

J. G48: CONICAL CAVITY MILLING CYCLE

Used for cutting straight conical, concave or convex radial shapes. A ball end mill has to be used in order to produce a radius. All dimensions are to the center of the cutter. The cutter must be at starting position X, Y and Z before programming this cycle. See figure 55.

The following variables must be programmed:

V60 Angle at which cone is to be cut. 3 o'clock is zero angle.

V62 Length of cylinder. Center of cutter to center of cutter.

V63 Number of cycles. One cycle is equal a cut in each direction across the cavity.

V64 Initial radius. Radius to be cut minus the radius of the cutter.

V65 Final radius. Radius to be cut minus cutter radius.

V66 Concave = 0 Convex = 1.

G48 Activates Cycle

The following program will cut the cavity show in figure 55 below, using a 3/8" ball end mill.

X 2.221 Y .779 Start position of cutter.

Z -.1875 Z depth minus radius of cutter.

V60: 45. Cone to be cut at 45 degrees counter clockwise.

V62: 2.000 Length of cylinder.

V63: 16. Number of cycles. This would take 32 across the cone; 16 in each direction.

V64: .3125 Initial radius. Radius to be produced is .5; radius of cutter is .1875. Therefore, initial radius equals $.5 - .1875 = .3125$

V65: .0625 Final radius. Radius to be produced is .25; radius of cutter is .1875. Therefore, final radius equals $.25 - .1875 = .0625$

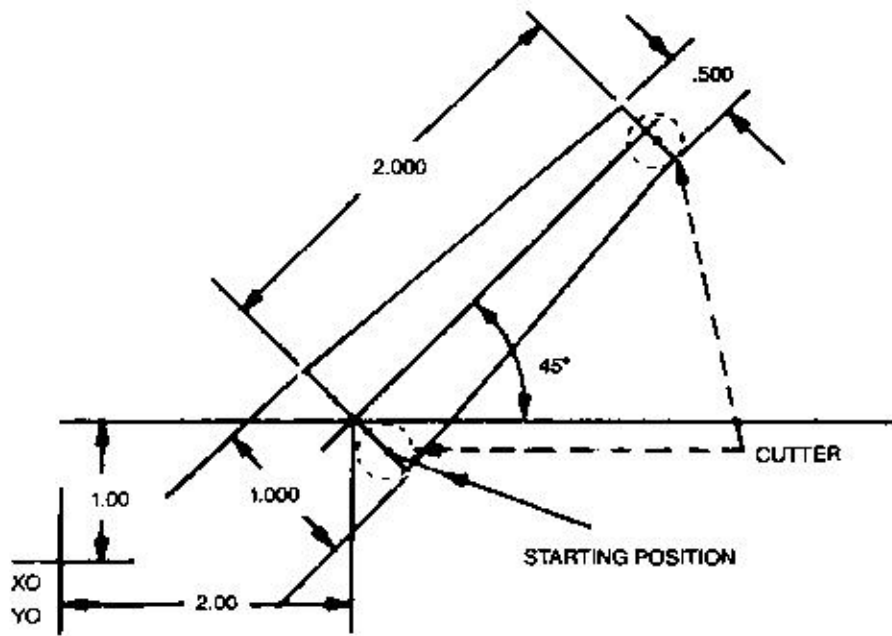


Figure 55

CONICAL MILLING CYCLE

V66 0. Concave cavity.
 G48 Activates cycle.

NOTE:

In order to produce a half circle the Z depth must be set down into the surface by the radius of the cutter.

4. G49: Radial Conical Milling Cycle

This cycle will cut a radial cone. The center of arc center around which cone is to be cut must be at X0 Y0. Cutter must be at starting point of cone. Cutter Z starting depth must be radius of cutter into part in order to produce full radius.

The following variable must be programmed to make this cycle work:

V62 Finish angle 3 o'clock is zero degrees.
 V63 Number of cycles. One cycle is 1 pass across in each direction.
 V64 Initial radius. Starting radius minus cutter radius.
 V65 Final radius. Finish radius minus cutter radius.
 V66 Concave 01 Convex 1.
 V75 Dimension of arc to be cut.
 G49 Activate cycle.

The following program will cut the radial cone in figure 15.

X 1.193 Y 1.193 Starting position of cutter.
 Z -.1675 Z depth is radius of cutter.
 V62 180 Finish angle. Angle of finish from 3 o'clock position.
 V63 16 Number of cycles. This will actually take 32 cuts, 16 in each direction across device.
 V64 .3125 Initial radius. Radius to be produced is radius of cutter .1675. Therefore, initial radius equal $.0 + .1675 = .3125$
 V65 .0625 Final radius. Radius to be produced is .25; radius of cutter is .1675. Therefore final radius equal $.25 - .1675 = .0625$.

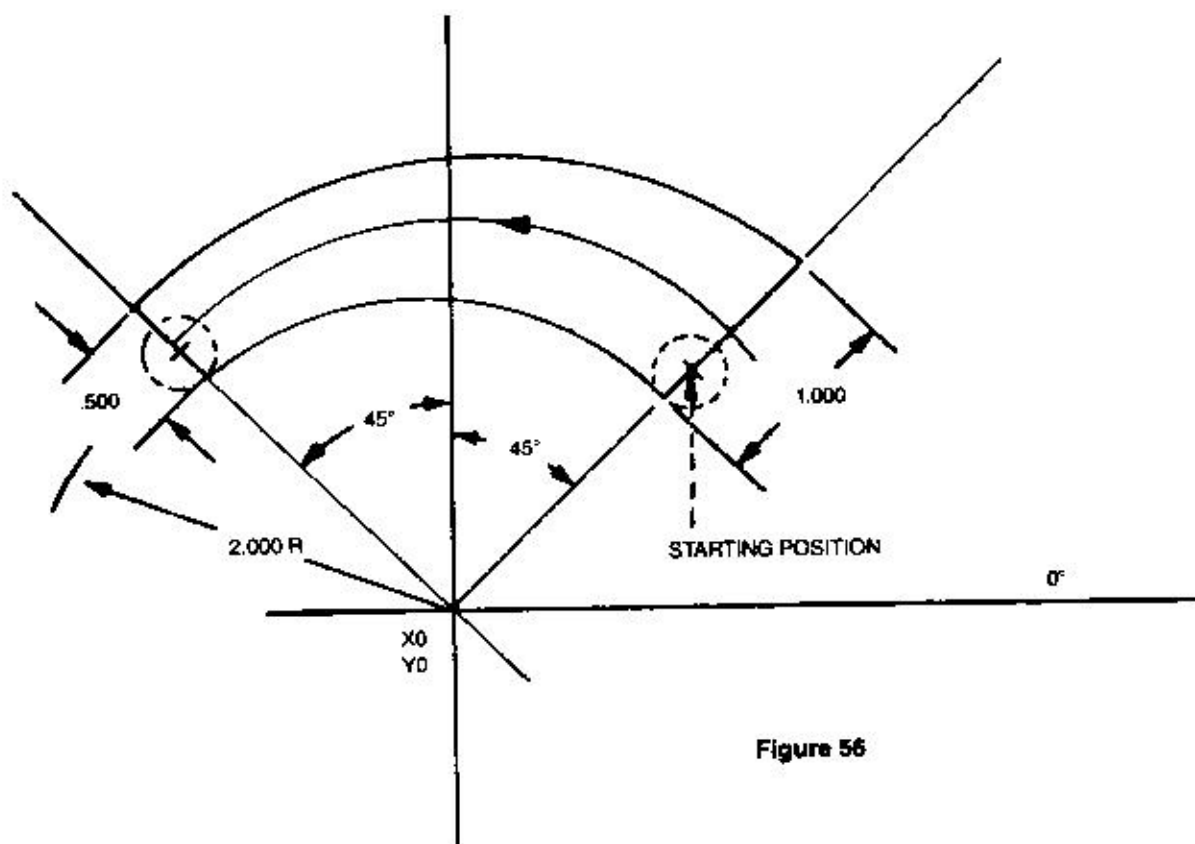


Figure 56

RADIAL CONICAL CAVITY MILLING

V66 0 Concave = 0.
V79 1. Cavity is to be cut in a counter clockwise rotation.
G49 Activated cycle.

L. G51: POLAR ROTATION

Polar Rotation is used when you need to rotate a program, or just a portion of a program, around a point. The operator can rotate holes, slots or milled shapes around a point.

For example, in Figure 57, it is necessary to mill 3 pockets as shown. Once the dimensions for pocket "A" are programmed, the operator can use Polar Rotation twice to rotate a pocket 120 degrees around the center point. This would eliminate the need to calculate and program the dimensions of pocket B and C. G51 causes every motion command to be rotated until a G52 is reached in the program. G51 works for X and/or Y moves only.

NOTE:

The first X Y move after G51 must be an absolute move.

The following variables must be entered before programming a G51:

V11 (Polar Center X)
V12 (Polar center Y)
V13 (Index angle of rotation)
V15 (Angle rotated)

V11 and V12 will describe the point which the program will rotate around. V13 describes the angle from which the program will rotate. Zero angle is 3 o'clock, but if program is written in such a way that the first subroutine is cut at 12 o'clock then V13 would still equal zero. V15 is the angle at which the subroutine is cut the second time. A plus angle rotates it in a counter clockwise direction and a minus angle rotates it in a clockwise direction.

In figure 57, the operator would put the dimensions of slot "A" in a subroutine (SUB. 1). The program to execute and rotate the slot to positions "B" and "C" would be as follows:

NOTE:

Cutter compensation must be turned off between rotations.

CALL 1 Execute SUB 1 position "A".
V11 1. Center X.
V12 1. Center Y.
V13 0. Index Angle.
V15 120 Angle of rotations: 120
G51 Activate Rotation

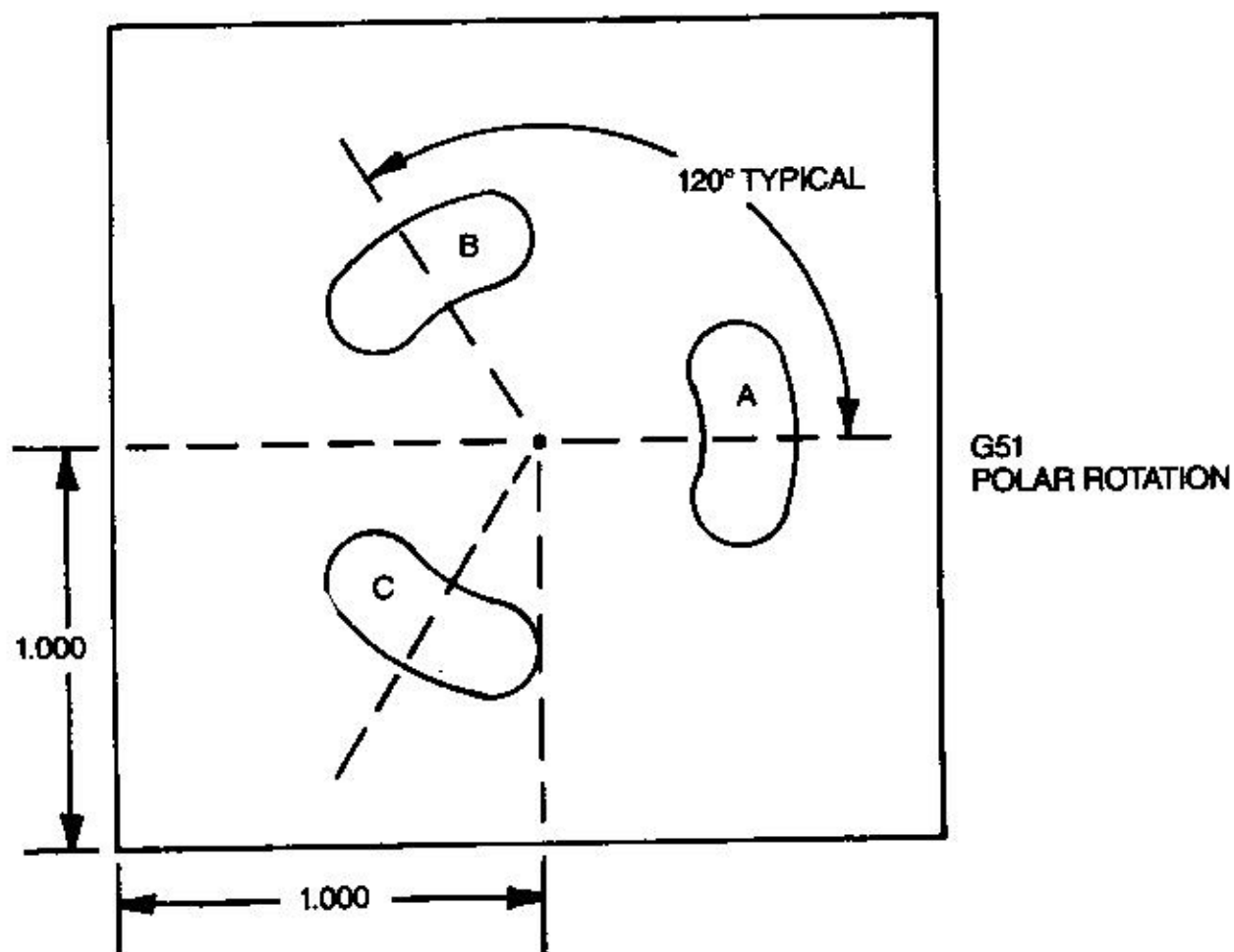


Figure 57

CALL 1		Executes SUB 1 position "B"
V11	1.	Center X
V12	1.	Center Y
V13	0	Index angle
V15	-120	Angle of rotation: -120
G51		Activtes rotation.
CALL 1		Executes SUB 1 position C.

G52

See figure 58. The drilled holes in this example are to be rotated 45 degrees from positions 1,2,3, and 4 to positions A,B,C,D. The program below will explain the procedure.

V20	10	Feedrate for drilling.
V21	.1	Drilling starting height.
V24	.1	Z retract height.
G81		Activate drilling canned cycle.
V11	1.	Center of rotation X.
V12	2.	Center of rotation Y.
V13	0	Index angle
V15	45.	Angle of rotation.
G51		Activates rotation.
X1.125 Y1.5 Z-.3 RA		XY coordinates and Z depth hole 1.
X.750	R1	Location of hole 2.
Y 1.	R1	Location of hole 3.
G52		Deactivate rotation.
G80		Deactivate drilling cycle.

NOTE:

In this example the 4 holes were to be rotated only once. It is not necessary to create a subroutine. If the 4 holes were to be rotated more than once, it would be wise to put the hole dimensions in a subroutine. That would reduce the programming steps.

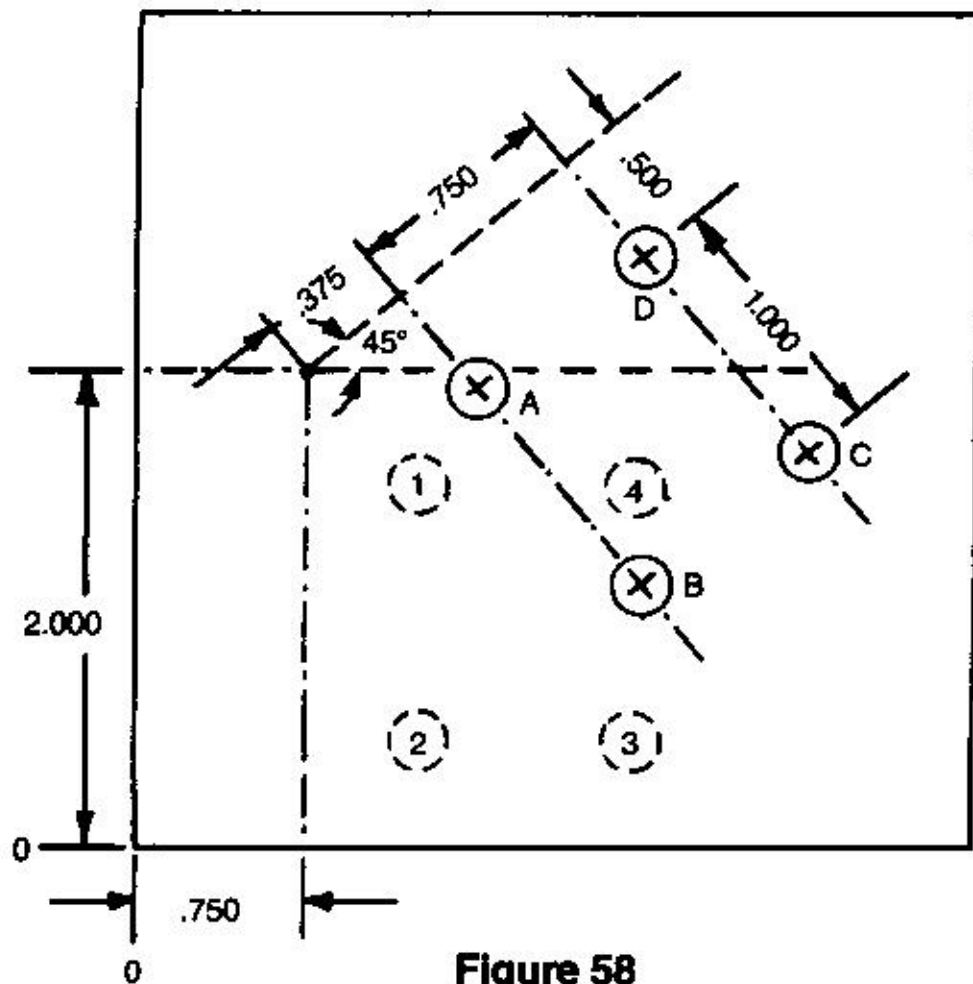


Figure 58

G51
2ND EXAMPLE
OF POLAR ROTATION

The program section written below for figure 59 shows how polar rotation can be used with a do loop to increment an angle. The curved slot (at the 3 o'clock position) will be put into a do loop to be repeated 7 times to produce 7 slots. Tool 1 is a .125 diameter end mill.

1. G52	Deactivate rotation.
2. T 1001	Tool 1 defined.
3. X Tool Diameter	Tool 1 offset.
4. Tool 0	Deactivate offsets.
5. Z0 RA	Retract Quill.
6. XD Y-3. RA	Rapid to tool change position.
7. T 1	Activate Tool 1 offset.
8. V11 2.250	Set center of rotation (X axis).
9. V12 1.	Set center of rotation (Y-axis).
10. V13 0	Set index angle.
11. V15 0	Set angle rotated (see note)
12. DO 7	Activates do loop repeat 7 times.
13. G51	Activates rotation.
14. X. 3.3125 Y1. RA	Rapid to end of slot.
15. Z.1 RA	Rapid Z to .1 above part.
16. Feed 10.	Set Feedrate.
17. Z -.05 FA	Feeds Z axis into part.
18. ARC CW	Start ARC clockwise
19. X.4375 Y0 FI	Defines ARC END point.
20. X.875 Y0 FI	Defines ARC end point.
21. ARC CW	ARC end statement.
22. Z.1	Rapids tool .1 above part.
23. V13 30.	Adjust V13 by 30.
24. END	End of do loop.
25. G52	Deactivates rotation.
26. Tool 0	Deactivates tool offsets.
27. Z0	Retract quill.
28. XD Y-3. RA	Rapids to tool change position.
29. END	End of program resets to event 1.

NOTE:

The do loop started with V13 and V15 is set to zero because the first slot is not rotated. Event 13 activates rotation but the slot is rotated 0 in the first loop.

Event 23 adjusts V13 to 30 degrees. Starting with the second loop, each slot is then rotated 30 degrees because G51 is activated.

Rotation can take place in the X and Y axis only. A Z axis move cannot be rotated.

G52: This G-code deactivates polar rotation

M. G53: SCALING

This feature allows the operator to scale the entire program or just a portion of a program, up and down to effect the size of the part being made.

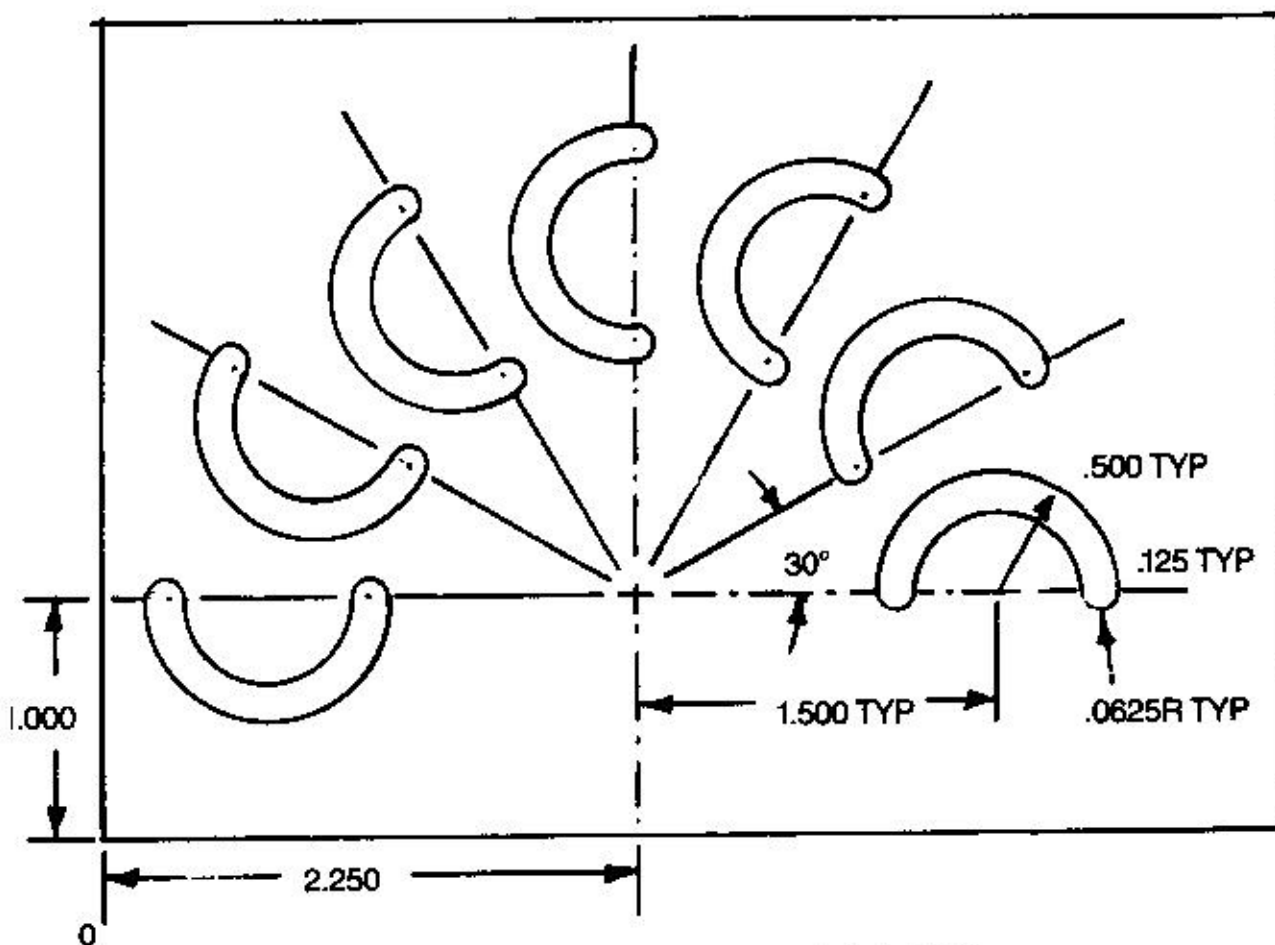


Figure 59: POLAR ROTATION

Part scaling is activated by G53. However, the following variables must be entered before G53 is programmed:

V11 (Scale center X)
V12 (Scale center Y)
V13 (Scale center Z)
V16 (Scale factor X)
V17 (Scale factor Y)
V18 (Scale factor Z)

Bear in mind that the first XY move after programming G53 code must be an absolute move.

Figure 60 shows the dimensions of a large box to be scaled down to smaller box.

The steps below show how this should be programmed into the Crusader Series-M.

1.	V11	0	Center X	
2.	V12	0	Center Y	
3.	V13	0	Center Z	
4.	V16	.5	Scale Factor X	
5.	V17	.5	Scale Factor Y	
6.	V18	1.	Scale Factor Z	
7.	G53		Activate Scaling	
8.	X0	Y0	FA	Tool starting point
9.	X	3.	FA	Original Coordinate
10.	Y	1.5	FA	Original Coordinate
11.	X	0	FA	
12.	Y	0	FA	
13.	G54			Deactivates scaling.

Figure 61, and the program steps below show how the operator may scale each individual axis with different scale factors. This example uses dimensions scaled from a certain point.

NOTE:

Scaling of circles is permitted if the scale factor for each axis is the same.

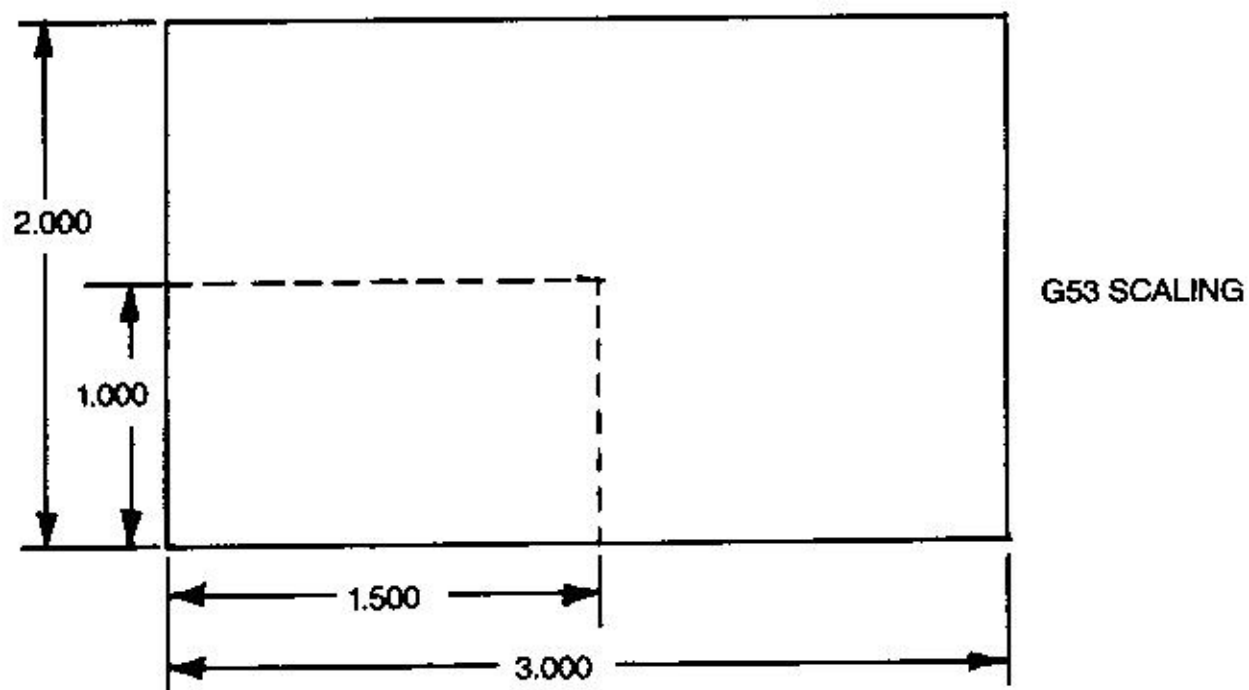


Figure 60

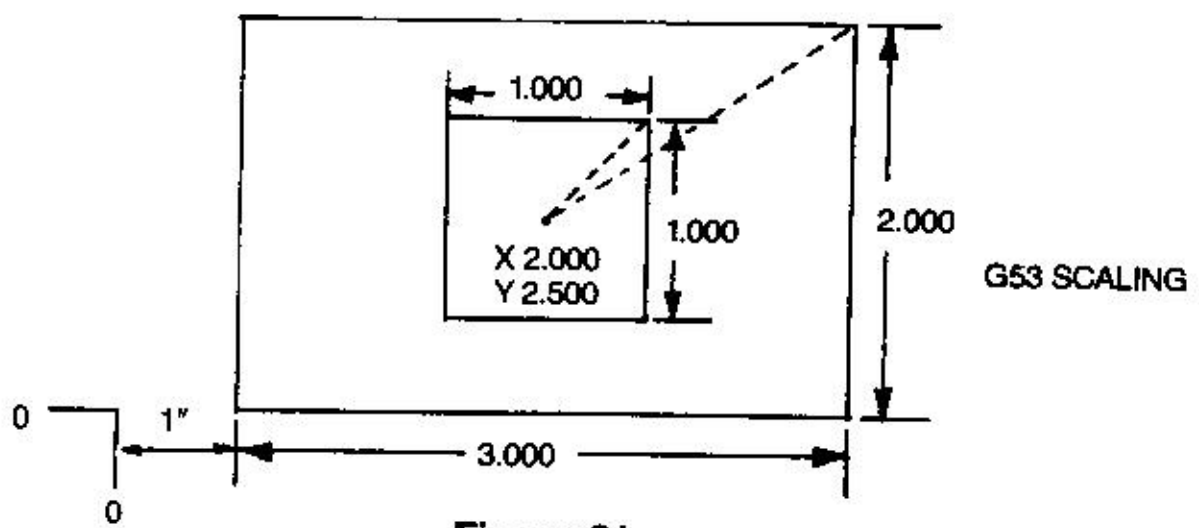


Figure 61

1. Feed	20.	Activates Feedrate
2. X 1.5 Y1.	RA	Move tool to center
3. V11	2.5	Center X.
4. V12	1.0	Center Y.
5. V13	0	Center Z.
6. V16	3.	Scale Factor X
7. V17	2.	Scale Factor Y
8. V18	1.	Scale Factor Z
9. G53		Activate Scaling
10. X 3. Y 1.5 FA		Original Start Point
11. Y-1.	FI	Original Coordinates
12. X-1.	FI	Original Coordinates
13. Y 1.	FI	Original Coordinates
14. X 1.	FI	Original Coordinates
15. G54		Deactivate Scaling

This program will take the 1 inch square box and make it a 3 x 2 inch rectangular box.

G54: This G-code will deactivate Scaling.

N. G75: FRAME MILLING

The programmer uses this canned cycle to mill a rectangular frame. The following variables must be entered before G75. (Activate frame milling cycle) is programmed.

V40	Z axis dimension .1 above the top surface of the part to be milled.
V41	The incremental X distance of the inside of the frame.
V42	The incremental Y distance of the inside of the frame.
V43	Absolute Z depth of frame.
V44	Corner radius on inside of frame
V45	Maximum incremental step over for each pass in X and Y.
V46	Maximum incremental Z pack depth per pass.
V47	Stock to be left on sides and bottom for finish pass.
V48	Feedrate for finish pass.
V50	Incremental width of frame.

The tool must first be positioned over lower left inside corner of frame. The program shown below will machine the frame. The shown in Figure 62.

1. X 2.000 Y 2.000 Z.100	RA	Rapid to start point.
2. Feed	10.	Roughing feedrate.
3. V40	.1	Starting height
4. V41	3.00	Length X.

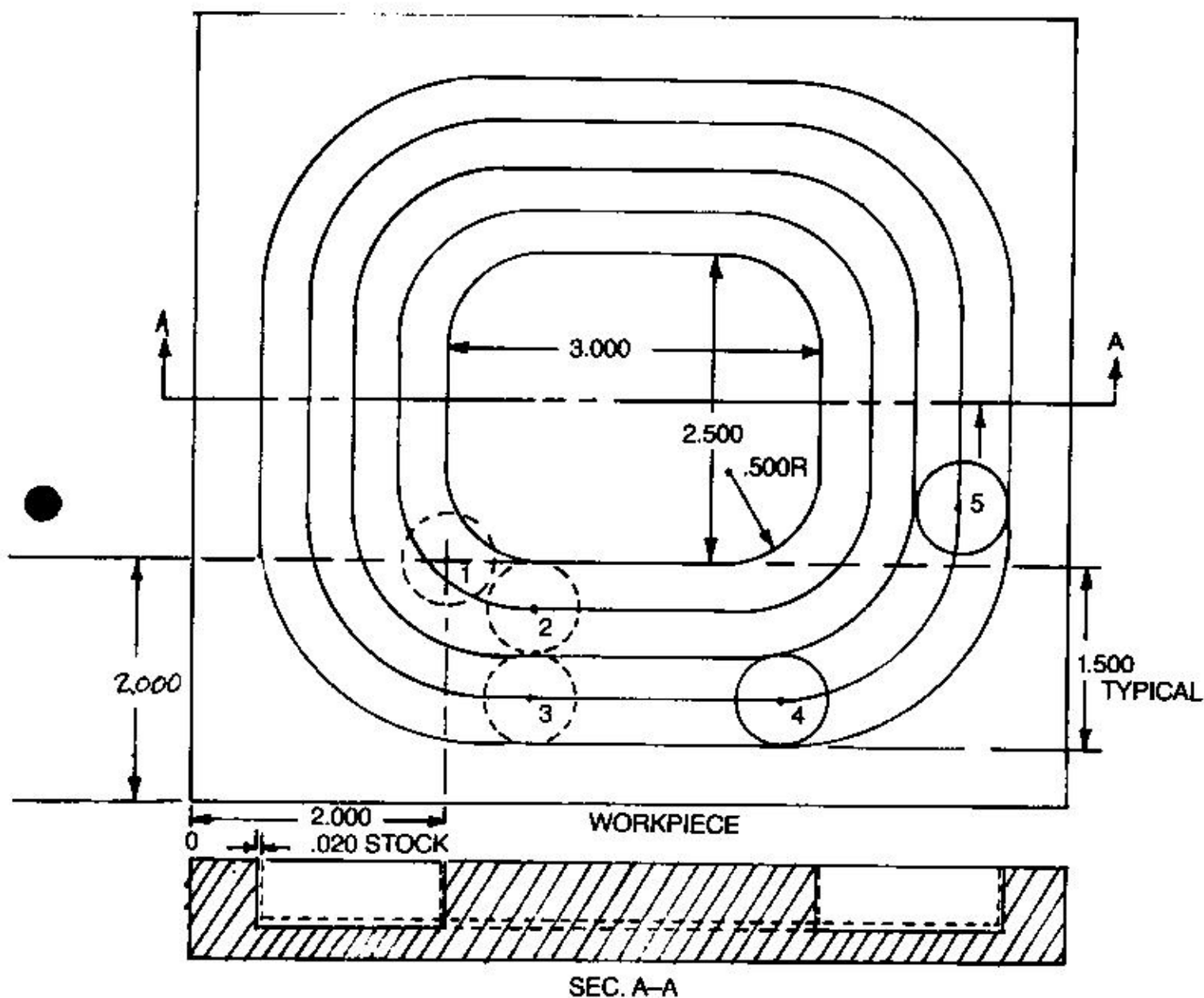


Figure 62

5. V42	2.5	Width Y.
6. V43	-.500	Depth Z
7. V44	.500	Corner Radius
8. V45	.500	Maximum X Y stepover
9. V46	.250	Maximum Z peck per pass
10. V47	.020	Finish Stock
11. V48	15.	Feedrate finish pass.
12. V50	1.5	Frame width
13. G75		Activates frame milling cycle.

The operator programs the tool to move to position 1. Upon execution of G75, the tool will rapid to V40 dimension, rapid to position 2, slope to position 3 at the first Z peck depth, feed to position 4 and 5 and continue around the frame back to position 3, feed to position 2 for the next pass around the part until the full width of the frame (minus the finish stock) is machined. The pattern is repeated until the finish depth (minus the finish stock) is reached. The finish pass is then made at the V48 finish feedrate.

0. G76: HOLE MILLING CYCLE

The hole milling cycle code (G76) can be used to mill a hole through a part or to do a counter bore. The operator must enter the following variable before G76 is programmed:

V18 (hole diameter).

The operator must position the tool at the center of the hole and have it placed at the required depth. When G76 is executed, the tool will follow the path shown in figure 63. The tool will first move 45 degrees, then approach the hole diameter on a tangent, contour 360 degrees, leave on a tangent, and then move back to center on a 45 degree angle.

To mill the hole shown in figure 63, put the tool in the center of the hole location and at the correct depth, then follow the program steps below.

V18	1.0000	Hole diameter
G76		Activates hole milling

P. G77: CIRCULAR POCKET MILLING

This cycle, which is activated by a G77 code, is used to mill circular pockets and eliminate extensive programming on the part of the operator.

The operator must enter the following variable before G77 is programmed:

V40	Z axis dimension .1 above the top surface of pocket area. This is an absolute dimension.
V41	Pocket center X.

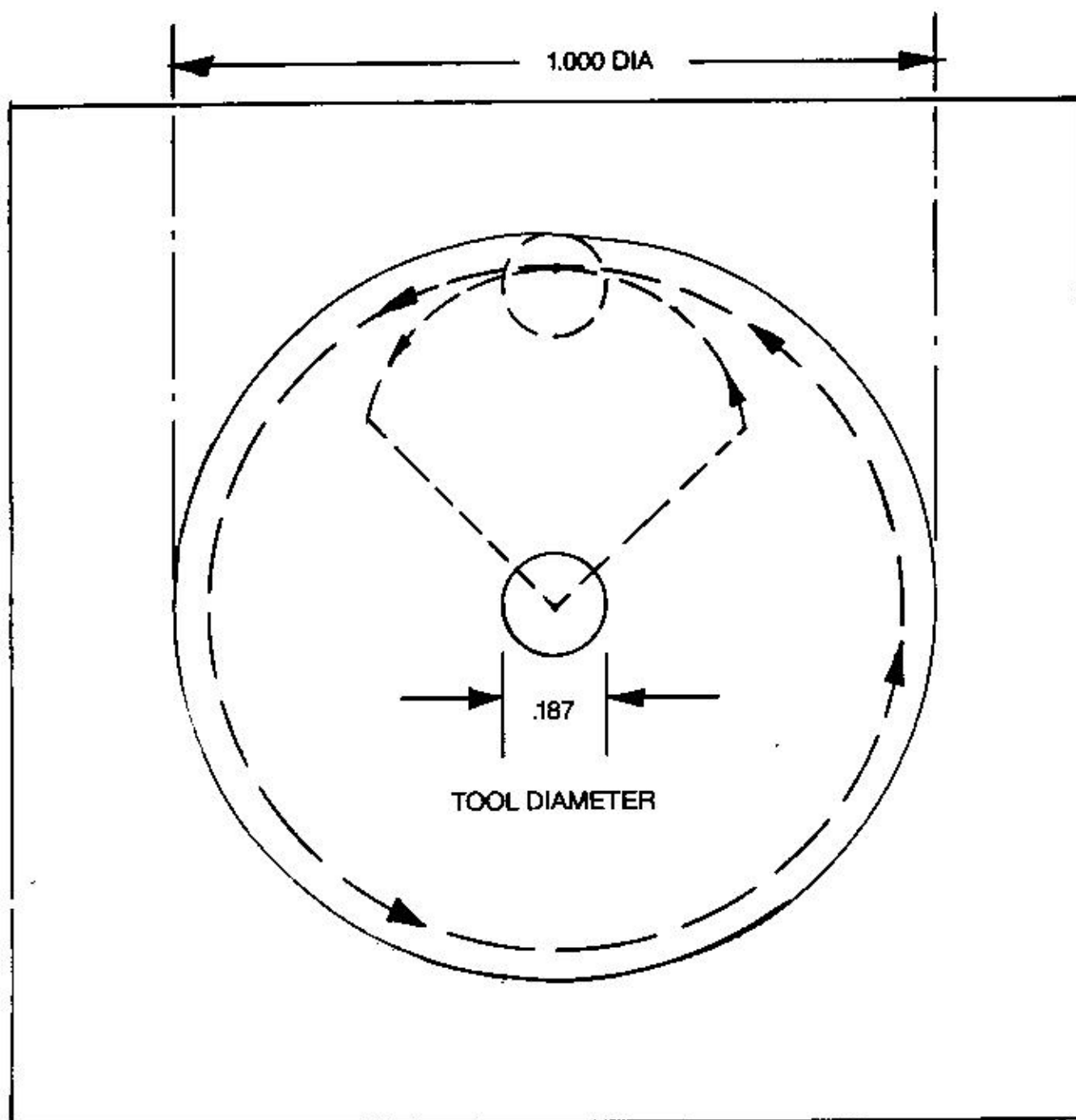


Figure 63: HOLE MILLING CYCLE (G76)

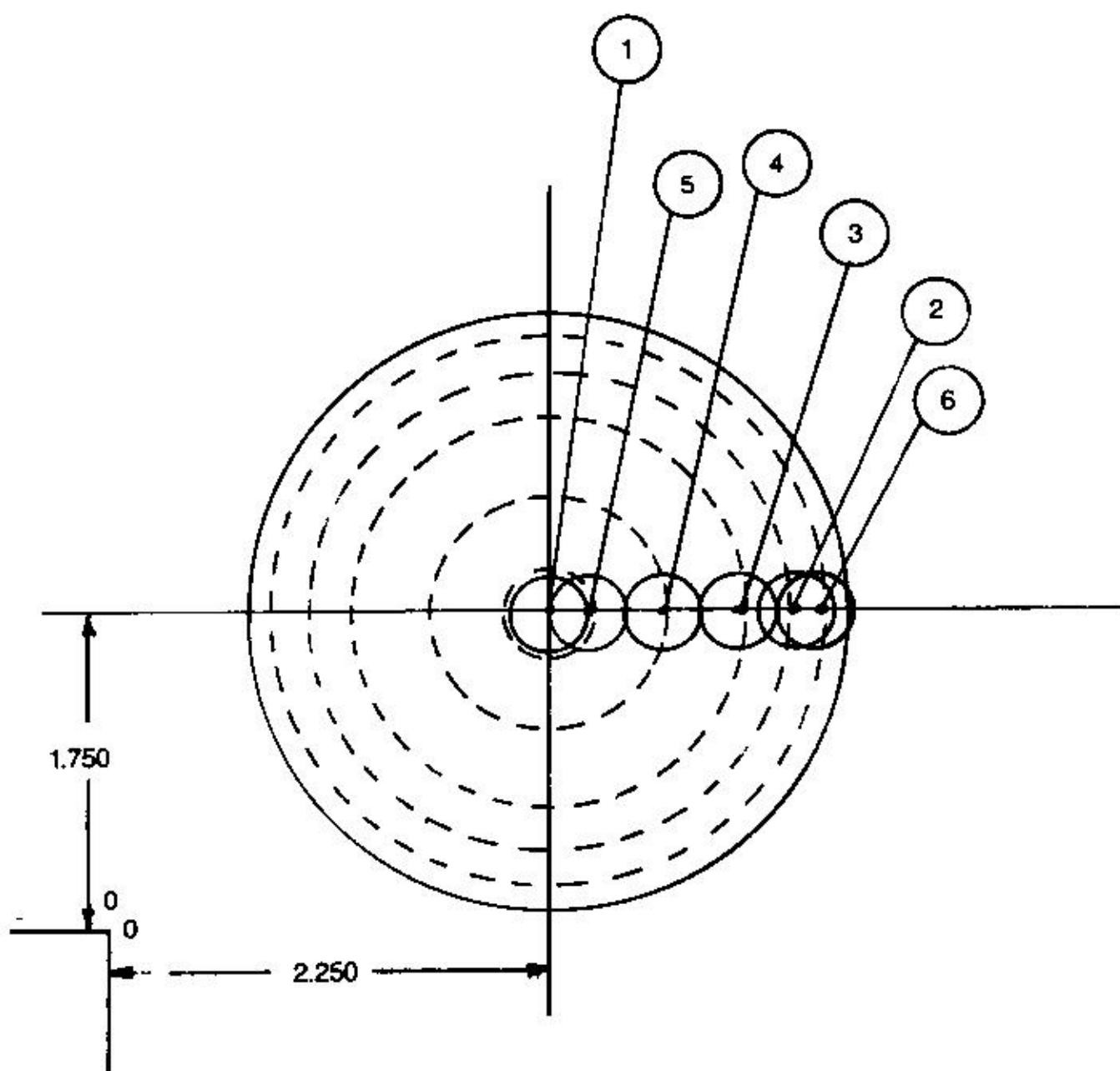
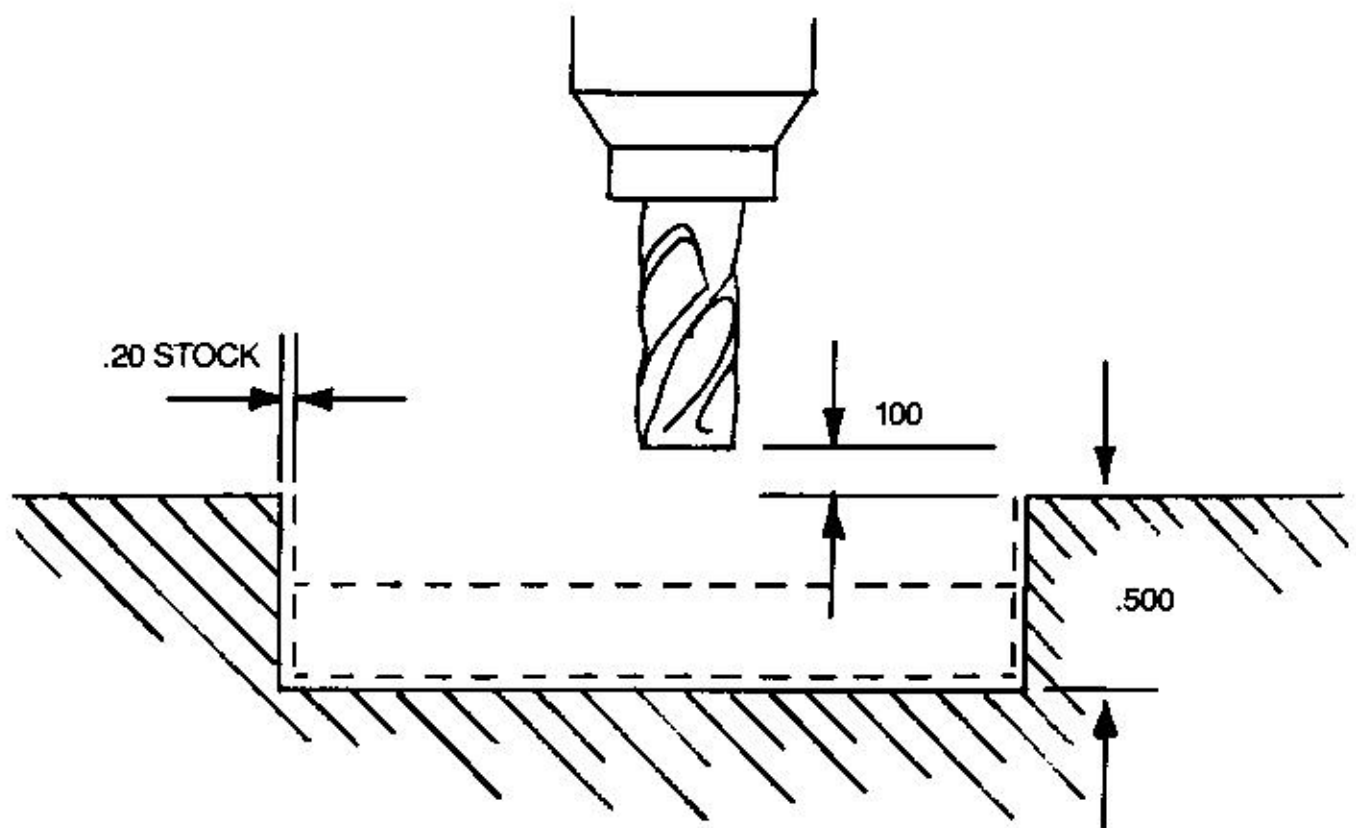


Figure 64: (G77)

POCKET 2.000 DIA (X 5 DEEP)

TOOL .250 DIA



V42 Pocket center Y.
 V43 Pocket depth Z absolute.
 V44 Pocket diameter.
 V45 Maximum incremental step over for each pass.
 V46 Maximum Z peck per pass.
 V47 Stock to be left for finish cut on sides and bottom.
 V48 Feedrate for finish cut, and V50 (0 if finish pass
 required and 1 if not required).
 V50 0 = Finish Pass; 1. = No Finish Pass

The program to cut pocket shown in figure 64 would be written as follows:

Feed	10.	Roughing feedrate
V40	.100	Starting height
V41	2.250	Pocket center X ABS
V42	1.750	Pocket center Y ABS
V43	-.5	Pocket center Z ABS
V44	2.	Pocket diameter
V45	.2	Maximum step over
V46	.25	Maximum Z depth per pass
V47	.020	Stock left for finish cut
V48	15.	Feedrate finish pass
V50	0	Finish cut required
G77		Activate circular pocket milling cycle

Upon execution, the tool will rapid to the X Y center (point 1) and rapid to the V40 dimension. It will then feed to top of work piece, slope to the edge of pocket (point 2) - minus finish stock at first depth. The tool will then contour 360 degrees, step over to point 3, contour 360 degrees until the center is reached.

If more roughing cuts are required, the slope-in and peck-over sequence is repeated. After the pocket is roughed out, the finish feedrate becomes active and the finish cut is taken on the side and bottom. The V40 dimension must be .100 (2mm) above the top of work piece.

NOTE:

The tool will pause for a few seconds upon execution of G77-G79 because the control is doing calculations. If an unprogrammed dwell occurs then an error has been made in the input data format.

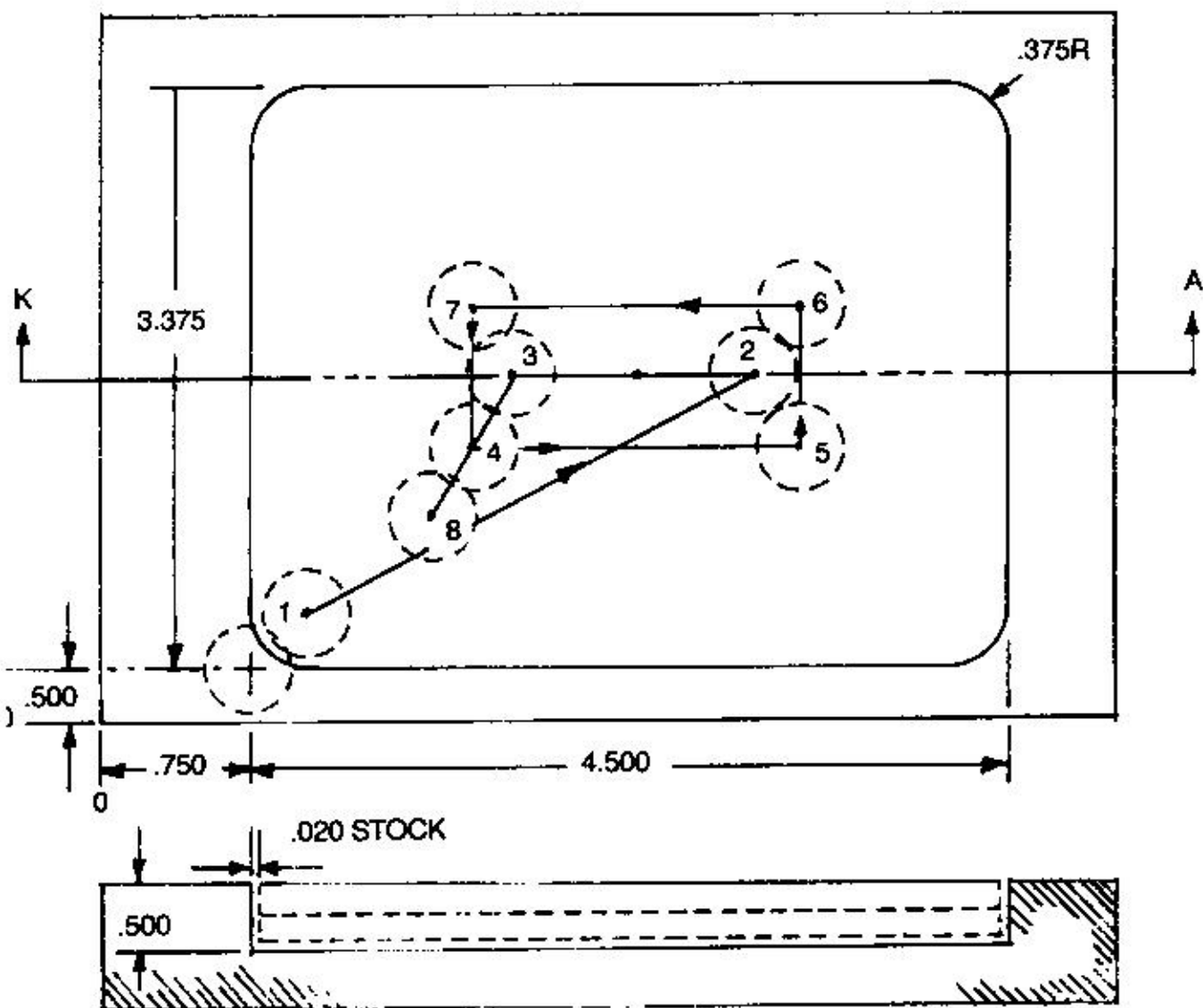
Q. G78: RECTANGULAR POCKET MILLING

The operator uses this cycle (activated by G78) to mill rectangular pockets without extensive programming. The operator must enter the following variables before programming the G78 code:

V40	Z axis dimension .1 above the top surface of the pocket.
V41	Incremental length in X.
V42	Y axis absolute depth of pocket.
V43	Absolute Z depth.
V44	Pocket corner radius.
V45	Incremental maximum step over for each pass in X and Y.
V46	Incremental maximum Z peck depth for each pass.
V47	Stock to be left on sides and bottom for finish pass.
V48	Finish pass feed rate.
V50	Set = to zero if a finish pass is required. Set = to 1., if no finish pass required.

The operator must first position the center of the tool at the lower left corner of the pocket. The program shown below will machine the pocket shown in figure 65:

.75	Y.5	RA	Position center of tool over lower left corner of pocket.
Feed	8.0		Rough feedrate
V40	.1		Starting height above pocket.
V41	4.5		Length in X axis (incremental)
V42	3.375		Width in Y axis (incremental)
V43	-.500		Depth in Z axis (absolute)
V44	.375		Corner radius
V45.	.350		Maximum X and Y step over
V46	.250		Maximum Z peck per pass



SEC. A-A
Figure 65

V47	.020	Finish pass stock.
V48	15.	Finish pass feedrate.
V50	0	Complete finish pass.
G78		Activates rectangular pocket milling cycle.

NOTE:

V45 (which is the step-over) should not be greater than 70% of the tool diameter or the tool will not clean all of the pocket. See figure 66.

The tool must first be positioned in lower left corner of pocket (position 1) upon execution of G78, the tool will rapid to the center of the corner radius (position 2). It will then rapid to the V40 dimension, feed to the top of the work piece, slope in Z axis to position 3 to the first peck depth. The tool then machines the pocket by moving to 4-3 until the tool is at the pocket edge (minus finish pass stock).

This sequence will be repeated until the final depth (minus the finish pass) is reached. The finish feedrate will then become active and the final finish pass taken. The tool will retract to the V40 dimension and move to the lower left corner that it started from.

If V45 is entered as a plus value, the tool will move in a climb mill motion for side of pocket. If V45 is entered as a negative value, the tool will move in a direct or conventional mill motion inside of pocket. The finish pass will always be a climb mill cut. The control will always adjust the maximum step move to make each pass equal. The Z peck will also be adjusted to make each peck equal. For instance, in this example, the maximum peck distance is .250, the total depth .500 and the finish stock .020. The control will make two 2 pecks at .240 each, then the finish stock .020 pass.

If a corner radius of .010 or less is calculated, a straight line move at 45 degrees will be made instead of a contour; otherwise the corner radius will be contoured automatically.

4. G79: BOLT HOLE CIRCLES

The bolt hole circle is programmed using the G79 code. To implement this feature, the operator must first enter a drilling cycle (G81, G82, G83, G85, G86, G87 or G89) and a Z depth to be drilled. The operator must then enter the following variables before programming a G79:

V11	Bolt circle center X.
V12	Bolt circle center Y.
V13	Index angle, shifted 0 angle from 3 o'clock position.
V15	Angle of first hole relative to 0 degrees.
V16	Angle of last hole.
V17	Number of holes to be drilled.
V18	Diameter of bolt circle.
G79	

The following program would center drill the pattern of holes shown in figure 67:

V20	10.	Drilling feedrate
V21	.100	Drill feed starting height
V24	.500	Drill rapid retract height
G81		Activate drilling cycle
	Z-.130 RA	Drilling depth
V11	2.	Center X
V12	.5	Center Y
V13	0	Index angle
V15	45.	Angle first hole
V16	-90. or 270.	Angle last hole
V17	6.	Number of holes
V18	2.250	Diameter of bolt circle
G79		Activates bolt circle routine
G80		Cancels drilling cycle

NOTE:

The tool will always move in a counter clockwise direction around the bolt circle, and the holes will always be equally spaced between the start angle and finish angle. The tool can be at any position when G79 is activated. Since G81 is active, the tool will not drill a hole at its present location when the Z depth is given, but will drill hole only when the next X and/or Y location is reached.

5. HELICAL INTERPOLATION

Helical interpolation is used to move the X and Y axis in a circular motion and the Z axis up or down in a straight line, all at the same time. Thread milling or fan-blade milling are among the most common uses for helical interpolation.

The program written below will move the tool from point A to point B in helical motion as shown in figure 68.

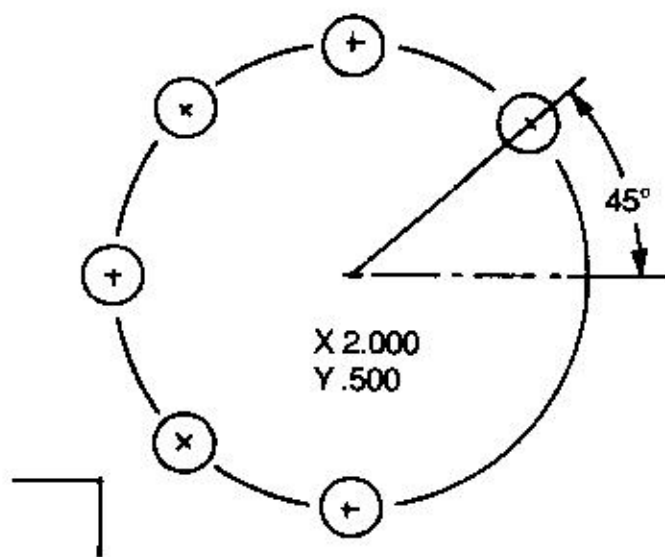


Figure 67

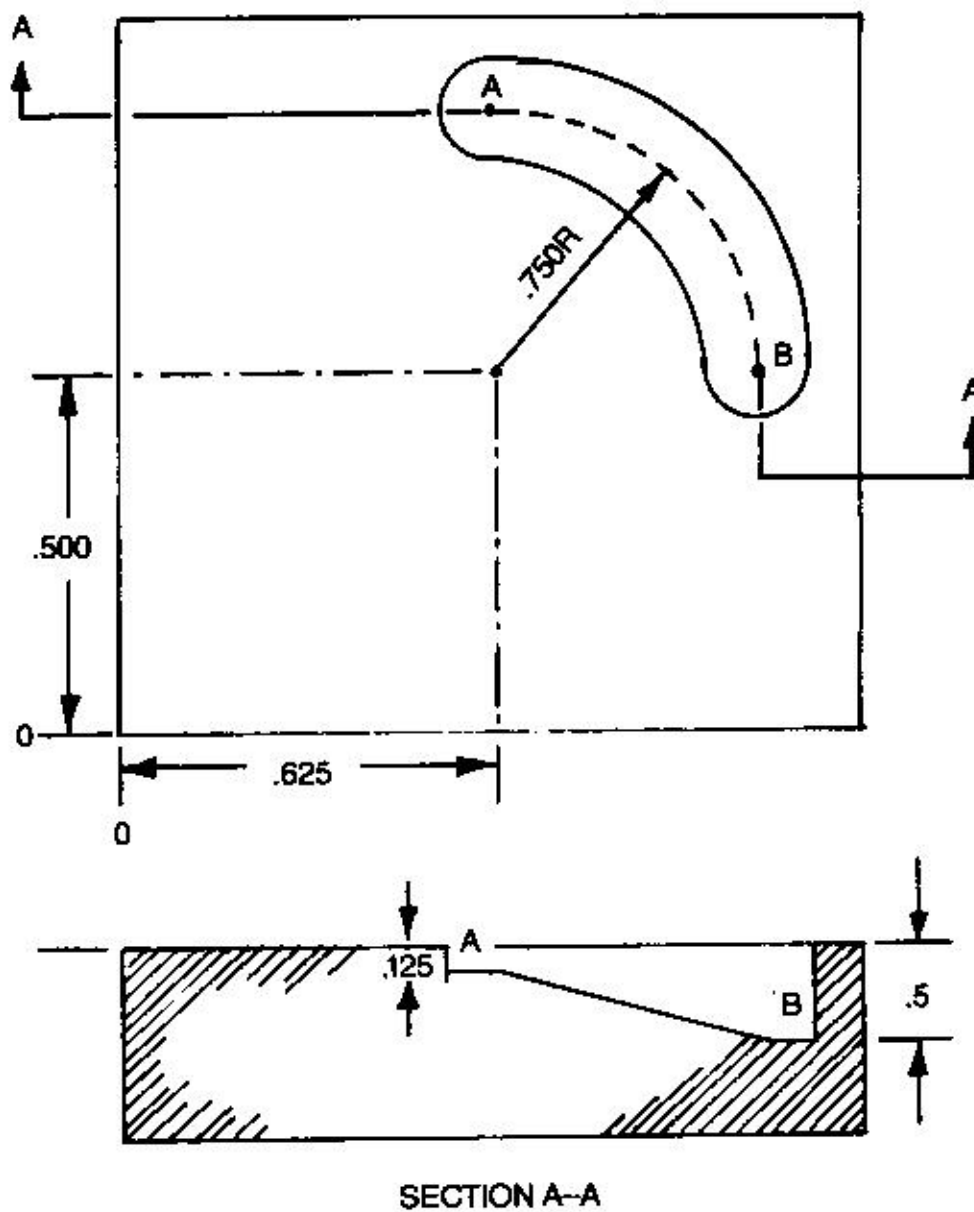


Figure 68: HELICAL INTERPOLATION

EVENT_No.:**FUNCTION:**

1. Tool 1001	Tool 1 offset definition event.
2. Z -1.123	Tool 1 offset.
3. Tool 0	Deactivate offsets.
4. Z 0 RA	Retract quill.
5. X 0 Y 0 RA	Rapid to tool change position.
6. Tool 1	Activate Tool 1's offset.
7. X.625 1.250 RA	Rapid to point A.
8. Z.05 RA	Rapid tool .05 above part.
9. Feed 5	Set feedrate.
10. Z-.125 FA	Feed tool to starting depth.
11. ARC CW	Describes arc direction.
12. X .625 Y .5 FA	Describes arc center.
13. V42 0.	Sets the number of complete 360 degree revolutions to be done. Since only 90 degrees is being cut in this example, V42 is set to zero.
14. X 1.375 Y .5 Z-.5 FA	Describes arc end point on all 3 axes.
15. ARC	Arc finish command.
16. Tool 0	Deactivates offsets.
17. Z 0 RA	Retracts quill.
18. X 0 Y 0	Returns to part change position.
19. END	Ends program.

In the next example (figure 69), the tool will move in helical motion to produce a thread with a thread milling cutter. Five threads per inch will be produced by starting and ending at the same X Y coordinate, setting the number of revolutions at 5 and moving the Z axis one inch.

1. Tool 1001	Tool 1 offset definition event.
2. Z -1.123	Tool offset.
3. Tool 0	Deactivate offsets.
4. Z 0 RA	Retracts quill.
5. X-2. Y 2.	Rapid to tool change position.
6. Tool 1	Activate Tool 1's offset.
7. X .5 Y 0 RA	Rapids to XY start position.
8. Z .05 RA	Rapids to .05 above part.
9. Feed 3.0	Sets feedrate.
10. Z 0 FA	Feed Z to top of part.
11. ARC CW	Describes arc direction.
12. X 0 Y 0 FA	Describes arc center.
13. V42 5.	Sets complete 360 degree revolutions to 5.
14. X .5 Y 0 Z -1. FA	Describes arc end point in all 3 axes.
15. ARC	Arc finish command.
16. Y -.5 FA	Feeds tool off part.
17. Tool 0	Deactivates tool offsets.
18. Z 0 FA	Retracts quill.
19. X -2. Y 2. RA	Rapids to tool change position.
20. END	Ends program.

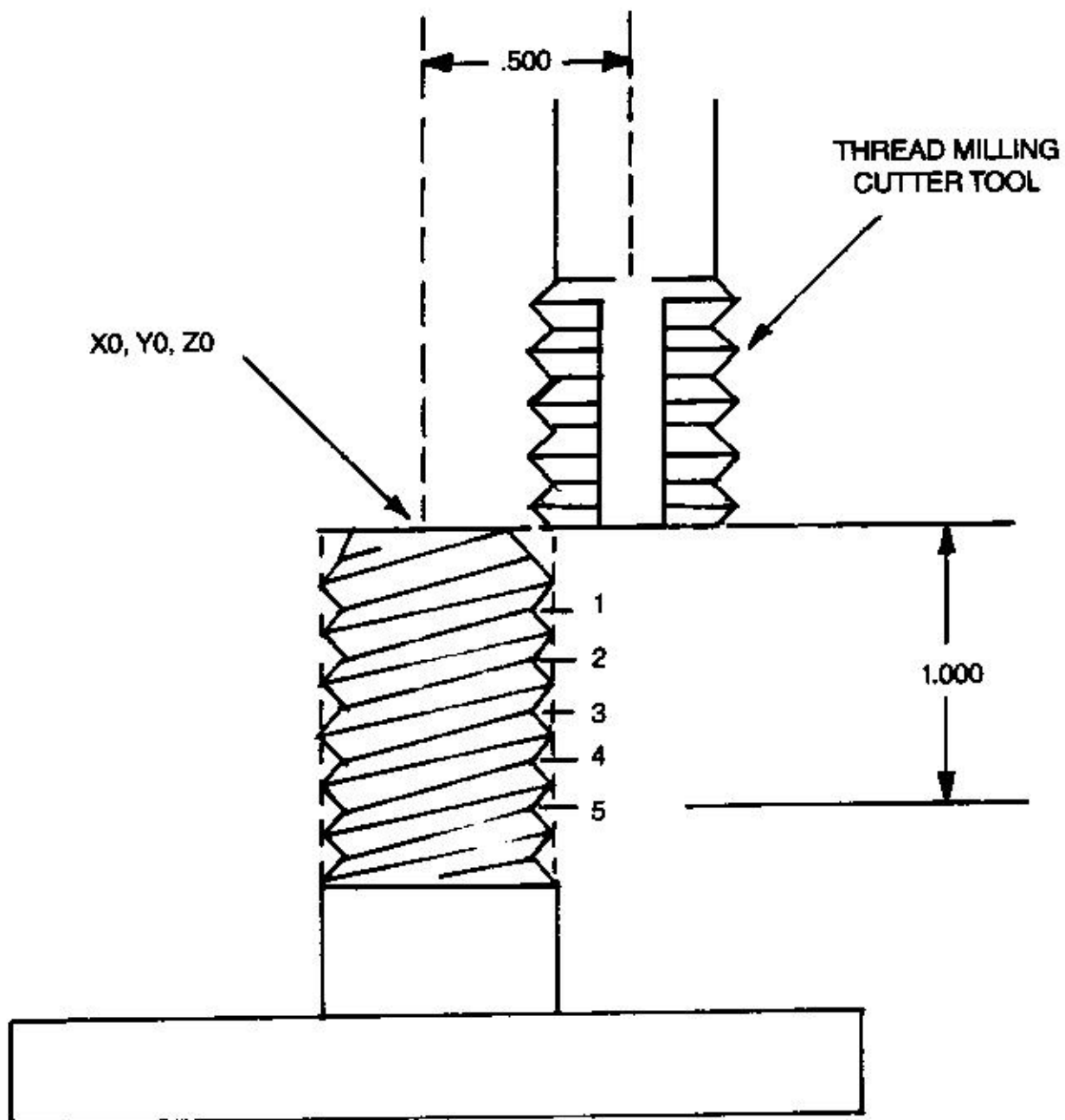


Figure 69: HELICAL INTERPOLATION

SECTION 12
UNDERSTANDING THE
HELP MENU

SECTION 12: UNDERSTANDING THE HELP MENU

A. WHEN TO USE HELP

Help can be used in two modes: MANUAL and PROGRAM ENTER. Help is entered by pressing the Help softkey indicated on the screen.

When help is pressed the screen will read as follows:

Push **G** for Help G code
Push **AUX** for Help AUX code
Push **Rapid** for Help RS-232
Push **ARC** for Help ARC and Helix
Push **T** for Help tool setting

You now choose which of the five you need help with.

B. G-CODE HELP

If **G** is pressed the screen will list the G codes in numerical order. When the softkey below the arrow pointing down is pressed the next page of G codes comes up on the screen. There are three pages total. Press the softkey under the arrow pointing down and the list will back up one page.

When you have determined which G code is required, then press that number (i.e. 05 gives you Ellipses). If in Manual the control will just list all the variables required in this canned cycle. In Program Enter it will ask for information one line at a time. Insert the necessary information, press **ENTER** and go to the next line.

In case you are not sure about what is required, press the **GRAF** softkey and the control will go to a graphic screen and show you what each V code is. The softkey that reads **GRAF** will now read **ENTRY**, so to get back to screen to enter information press this softkey again.

When all information has been inserted, press **ENTER** and this will put all variables and G codes into the program. If an error has been made in entering a V code you can back up through the V codes by pressing **PREV**. When you get to the one that needs to be changed press **DELETE** and enter the new information. Now, press **ENTER**. To get back to the V code you were at, press **NEXT** until that one is reached.

When adding a canned cycle into a program it is not necessary to open a gap in the program. This is done automatically by the control.

C. AUX CODE HELP

If AUX is pressed while in Help mode the control will list all AUX functions. With some of these functions the control has to be in Manual mode in order to enter them. All AUX codes will be discussed in next section. Page over as previously described using softkeys below arrow.

D. RS-232 HELP

While in help mode, press RAPID. This will list the AUX codes to set up for RS-232 (AUX 2700 series). These are entered into control in Manual mode.

E. ARC HELP

Press either ARC CW or ARC CCW to access this help section. You now will have to select the type of arc required (1) clockwise Arc, (2) counter clockwise arc, (3) helix clockwise, or (4) helix counter clockwise. These can be entered into the program in same way as G codes.

SECTION 13
AUXILIARY CODES

SECTION 13: AUXILIARY CODES

A. AUXILIARY CODES

The AUX button is used in the main program to cause variations in the standard control functions. These variations are assigned numbers and are entered into the program prefixed by the AUX button (i.e. AUX 0007, AUX 0400, AUX 2500, etc.). Below is a listing of all AUX codes and their functions.

AUX 0001	See OEM manual
AUX 0002	See OEM manual
AUX 0003	Spindle CW
AUX 0004	Spindle CCW
AUX 0005	Spindle OFF
AUX 0006	See OEM manual
AUX 0007	Coolant ON
AUX 0008	Coolant OFF
AUX 0009	See OEM manual
AUX 0010 -0099	See OEM manual
AUX 0100	Mirror X
AUX 0200	Mirror Y
AUX 0300	Mirror X and Y
AUX 0400	Mirror Z
AUX 0500	Mirror X and Z
AUX 0600	Mirror Y and Z
AUX 0700	Mirror X, Y and Z
AUX 0800	Deactivate mirror
AUX 1000	No deceleration (contouring)
AUX 1101	Zero shift
AUX 1110	Outer software limits OFF
AUX 1112	Enable outer software limit
AUX 1113	Set outer software limits
AUX 1114	Disable inner software limit
AUX 1116	Enable inner software limit
AUX 1117	Set inner software limit
AUX 1150	Disable shifted outer software limit
AUX 1152	Enable shifted outer software limit
AUX 1153	Set shifted outer software limit
AUX 1154	Disable shifted inner software limit
AUX 1156	Enable shifted inner software limit
AUX 1157	Set shifted inner software limit
AUX 1400	Feedrate override
AUX 1401	Feed and rapid override
AUX 1410	Cancel vector rapid mode
AUX 1411	Set vector rapid mode
AUX 1420	Clear Z and feed inhibit
AUX 1421	Set Z move inhibit
AUX 1422	Clear Z move inhibit
AUX 1423	Set feed inhibit
AUX 1424	Clear feed inhibit
AUX 1425	Set Z and feed inhibit
AUX 1500	Unlock program enter
AUX 1501	Lock program enter

(Listing is continued on next page...)

AUX 1603	Simulation off
AUX 1604	Simulation on
AUX 1605	Beeper OFF
AUX 1606	Beeper ON
AUX 1608	Display free memory
AUX 1900	Single step one event
AUX 1901	Single step one move
AUX 2000	Contouring OFF (cancel AUX 1000)
AUX 9030	Cancel plane rotation
AUX 9031	Plane rotation around X
AUX 9032	Plane rotation around Y
AUX 9033	Plane rotation around Z

In the following sections, the AUX codes will be explained in more detail as necessary.

NOTE:

This note is for AUX 9031, 9032 and 9033. When any of the AUX codes listed above are being used and are interrupted, then before trying to move the machine in Manual mode, you must first cancel Rotation. To do this, select MANUAL mode; press AUX 9030 and START. The machine can now be moved manually with no problems.

B. MIRROR IMAGE AUX CODES

Aux 100 - 800 affect the Mirror Image feature of the control. These mirror image AUX codes will reverse the plus or minus signs of the programmed dimensions to produce an opposite part.

A mirror image will stay active until deactivated by AUX 800 (see list) or another mirror image. The hole patterns and milled shapes in figures 70, 71 and 72 could be programmed using mirror image.

- AUX 100: Reverses the signs of the X axis dimensions.
- AUX 200: Reverses the signs of the Y axis dimensions.
- AUX 300: Reverses the signs of the X and Y axis dimensions.
- AUX 400: Reverses the signs of the Z axis dimensions.
- AUX 500: Reverses the signs of the X and Z axis dimensions.
- AUX 600: Reverses the signs of the Y and Z axis dimensions.
- AUX 700: Reverses the signs of the X, Y and Z axis dimensions.
- AUX 800: Turns the mirror image off. Entered into program as an event.

C. AUX 1000: CONTINUOUS PATH.

This should only be used for contouring continuous tangent lines

AUXILIARY MIRROR IMAGE

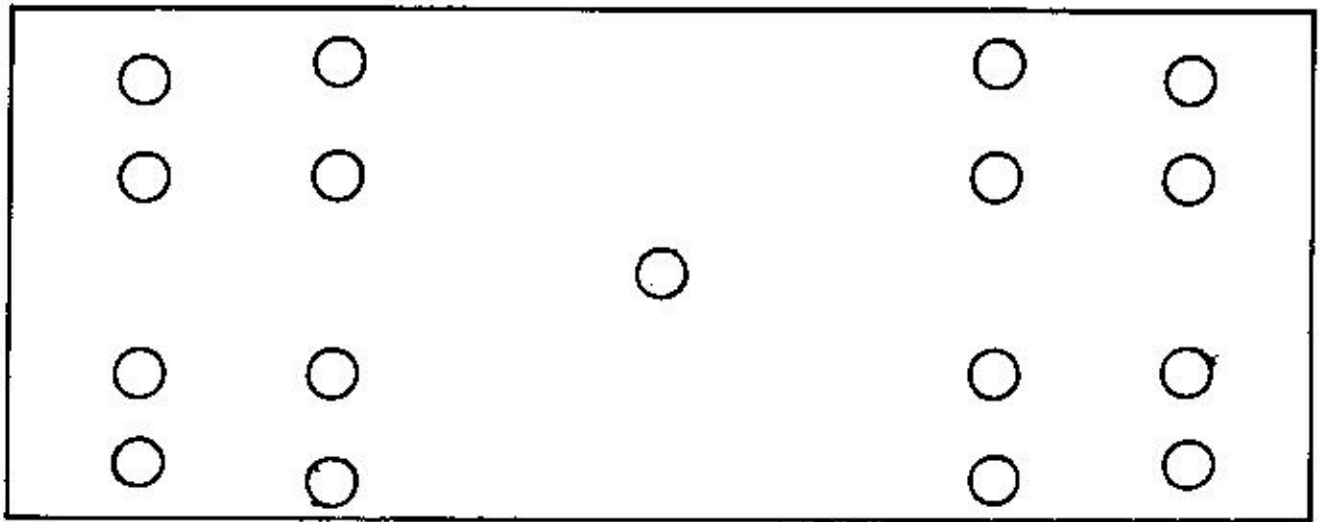


Figure 70

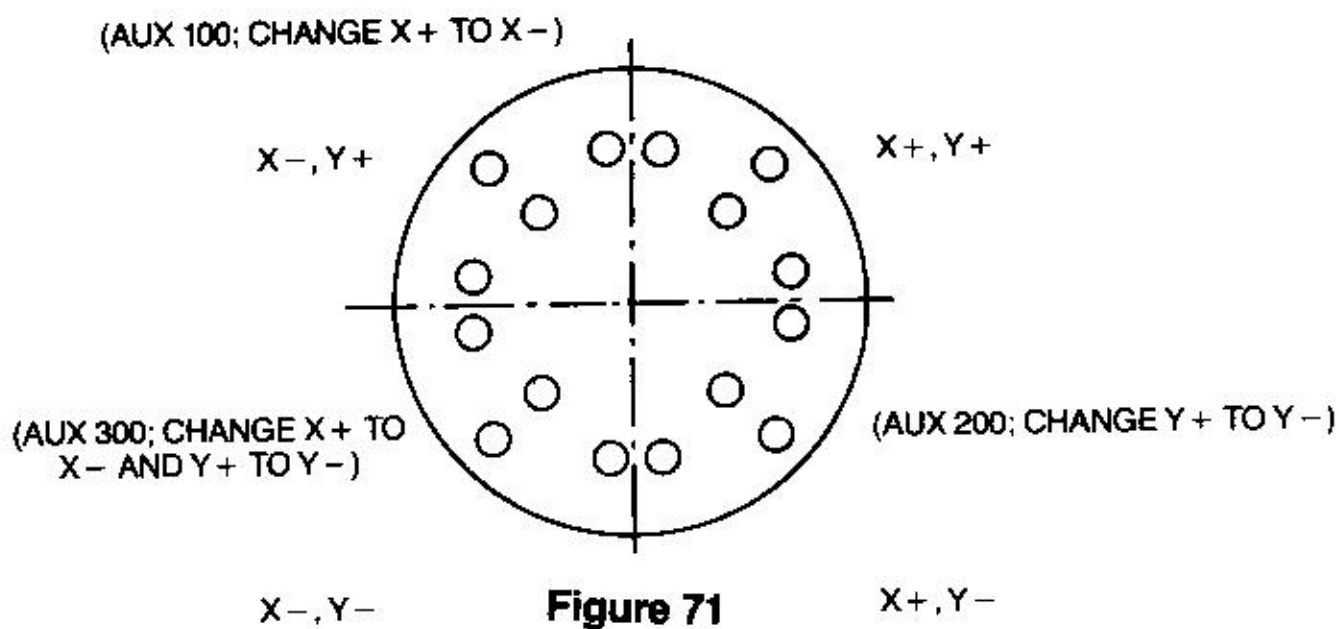


Figure 71

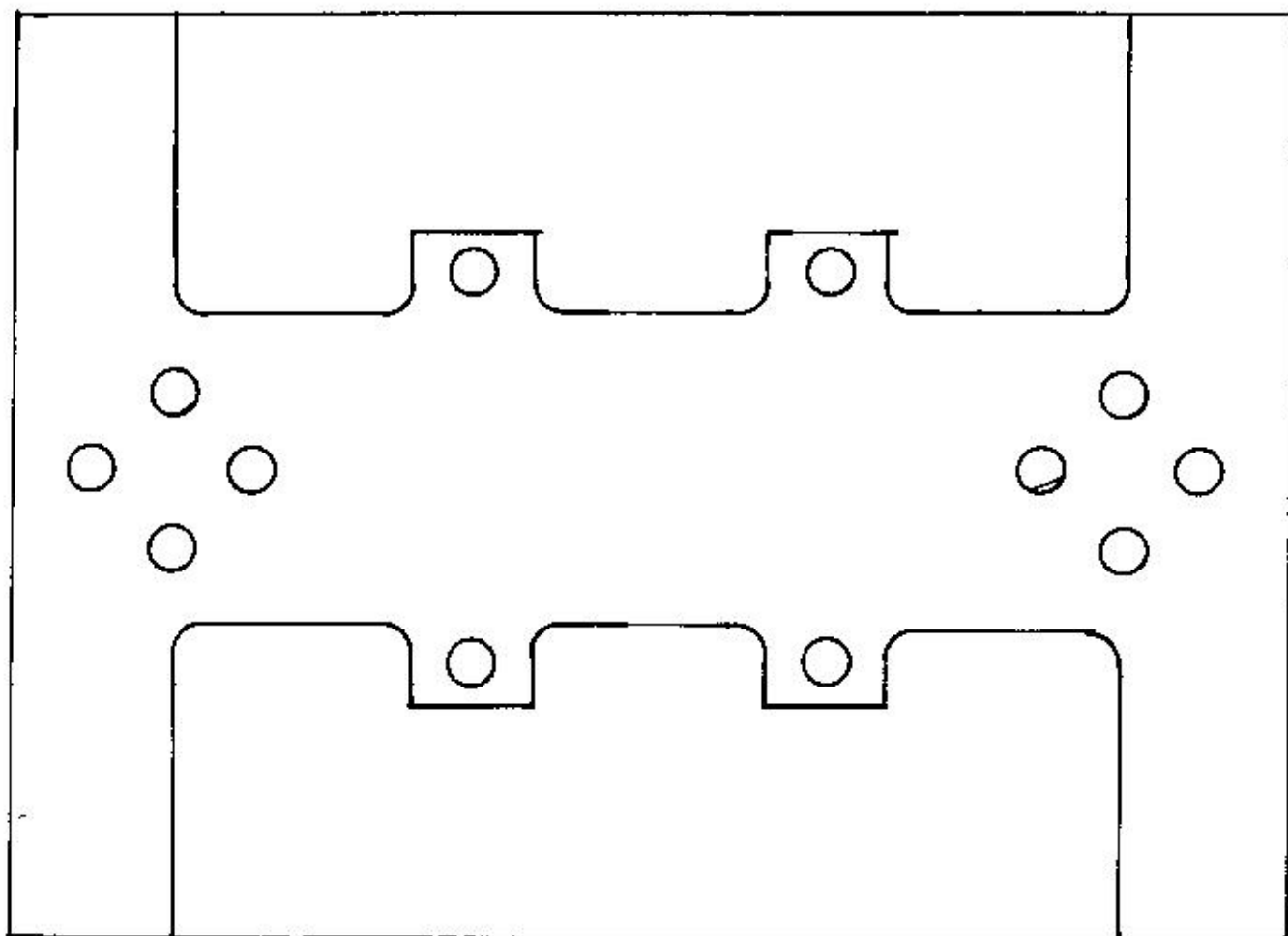


Figure 72: AUXILIARY MIRROR IMAGE

and circles and should be turned off immediately after use. It is not to be used in rapid or when doing consecutive straight line moves or over 40 inches per minute in feed. This command is modal and becomes active for next motion command. It causes pre-processing to take place, so the machine doesn't pause between moves but instead executes continuous motion. It is turned off with AUX 2000 and is automatically entered into the program as an event.

D. AUX 1101: ABSOLUTE ZERO SHIFT

This feature allows the programmer to move absolute zero from point to point on the table. When the operator is using several vises or is holding several workpieces in a large fixture, then the Absolute Zero Shift feature is useful. The operator only has to program the part once, then after each program is complete on one part, he simply shifts the absolute zero point to the next workpiece and runs the program again (figure 73).

AUX 1101 followed by X 3. Y 0 (tool length offset currently being used) will shift absolute zero to those X, Y and Z coordinates. Z is the tool length offset. To shift the zero back to the original absolute zero, program AUX1101, then X0, Y0, Z0. All zero shifts are measured from the original absolute zero. The zero shift information must be restated after each tool change, including changing TOOL 0.

E. AUX CODES FOR SOFTWARE LIMIT SETTINGS

Software limits allow the programmer to define a "box" from which the tool cannot move out. Collisions between vises, fixtures and tooling, along with damage to the machine itself, can be avoided. Once these limits are set and activated, the tool will not move past them in any mode. Software limits can be used as a safety stop to avoid drilling into the machine table, fixture or parallels. If a part overhangs the table, limits can be set to stop it from crashing into the column of the machine. If the operator tries to move past the limits set, the drive motors will cease to move the machine.

AUX 1110: Disables outer software limits

AUX 1113: Set outer software limits. This forms a "box" from which the spindle will not move out while limits are active. The tool must be inside box before limits are activated. All dimensions are absolute from X0, Y0.

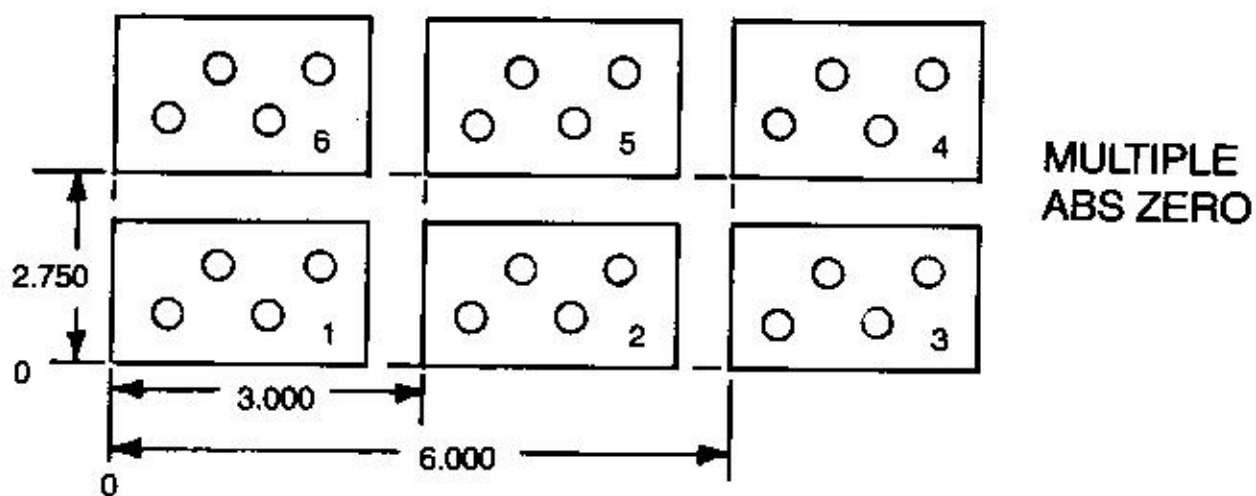


Figure 73

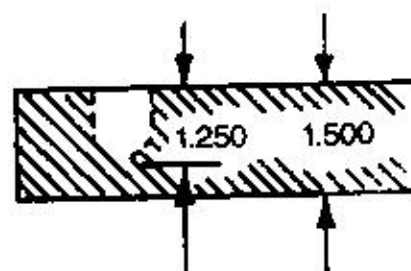
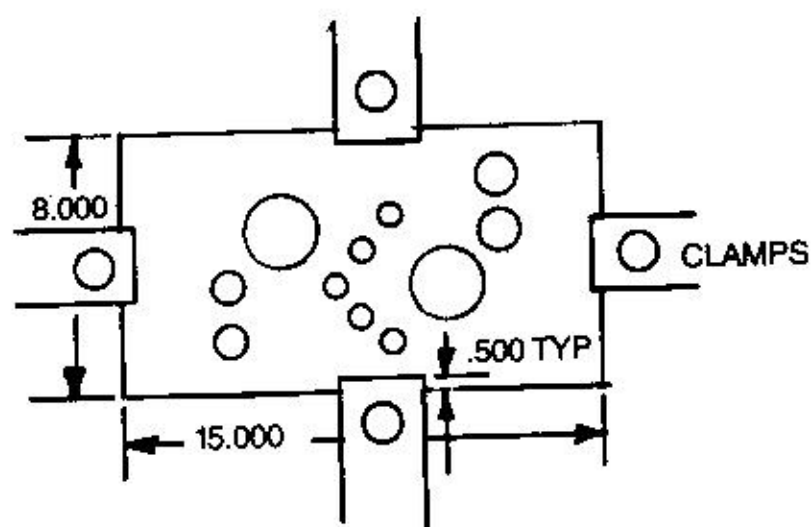


Figure 74 : SOFTWARE LIMIT

The following variables must be included when programming with:
AUX 1113:

V01 (-X)
V02 (+X)
V03 (-Y)
V04 (+Y)
V05 (-Z)
V06 (+Z)

The minus dimension can be programmed as a plus dimension. See the programming example and figure 74 below:

V01 .5
V02 1.5
V03 .5
V04 1.5
V05 -1.
V06 1.

While software limits are active, the center of the spindle on X and Y will not work outside the "box".

- AUX 1112: Activates software limits previously established by AUX 1113. Allows operator to activate whenever necessary.
- AUX 1114: Disables inner software limits.
- AUX 1117: Sets inner software limits. Works the same as AUX 1113 except now center line of the spindle on X & Y will not go INSIDE the shaded area while active.
- AUX 1116: Activates inner software limits. The same as AUX 1112.
- AUX 1150, 1153 and 1152: These work the same as AUX 1110, 1113 and 1112 but are used when AUX 1101 (absolute zero shift) is in effect. The same dimension would be used as in AUX 1113.
- AUX 1154 1157 and 1156: These work the same as AUX 1114, 1117 and 1116.

F. AUX CODES FOR BACKLASH COMPENSATION

- AUX 1160: Deactivate backlash compensation
- AUX 1161: Sets and enables backlash compensation. The

following variables must be set before AUX 1161 is programmed.

V02: Amount of backlash X
V04: Amount of backlash Y
V06: Amount of backlash Z

The values should all be entered as plus dimensions. This AUX code is modal, therefore once set and enabled it does not need to be entered on every program. After entering these values into machine a move in the plus direction should be made. This sets the control so that it will compensate any moves thereafter.

This AUX code is entered in Manual and retained until changed.

AUX 1162: Reactivate backlash compensation. If backlash compensation had been turned off this AUX code will switch it back on.

G. AUX 1400: FEEDRATE OVERRIDE

FEED % override for feed moves only. Rapid moves will be made at normal rapid rate. This AUX code is active at start up. This command is modal.

H. AUX 1401: FEED AND RAPID OVERRIDE

When this AUX code is entered into the control both feed and rapid can be controlled by FEED % buttons. This command is modal.

I. AUX 1410: CANCEL VECTOR RAPID

Causes rapid motion to be executed in a straight line. This is a modal command and is active at start up.

J. AUX 1411: SET VECTOR RAPID

Allows all axis to run at maximum rapid speed when a rapid move is programmed. The tool will not travel in a straight line when this AUX code is active. This is a modal command.

K. AUX 1500 and 1501: PROGRAM ENTER LOCK

These AUX codes allow us to protect a program that has been entered into control. First control must be in manual mode and Area C line 2 has to read COMMAND (CNMD). Press V 8 9, enter a number (any number) and press START.

Next, press AUX 1 5 0 1 START. Program Enter mode is now locked out and program can not be changed.

To unlock, reverse this procedure and use AUX 1500.

L. AUX 1605 and 1606: BEEPER OFF/ON

AUX 1605 turns beeper off, AUX 1606 turns beeper on. This is

done in the Manual mode. Press MANUAL AUX 1 6 0 5 START and beeper is no longer active. The same process is used to turn it on except AUX 1606 is entered into control. Default at start up is beeper ON.

M. AUX 1608: DISPLAY FREE MEMORY

This AUX code allows you to see how many more events you have left to use, after entering the program. Press MANUAL AUX 1 6 0 8 START. The control, after a short period of time, will display a message in Area D that reads FREE MEMORY BLOCK: XXXX

N. AUX 1900: SINGLE STEP (EVENT)

Single Step mode will execute one event each time the Start button is pressed.

O. AUX 1901: SINGLE STEP (MOVE)

This mode will execute one X Y or Z move each time the START button is pressed (active at power up).

P. AUX 2000: Cancels AUX 1000 (active at power up)

Q. AUX 9030: CANCELS PLANE ROTATION

Used to cancel AUX's 9031, 9032 and 9033

R. AUX 9031, 9032 and 9033: Plane rotation

Aux 9031: Rotates around X axis
Aux 9032: Rotates around Y axis
Aux 9033: Rotates around Z axis

These AUX codes are used to rotate a profile or shape around an axis. Point from which it is rotating must be 0. Shape or profile must be programmed incrementally. Cutter compensation cannot be used with plane rotation.

The following shape (figure 75) was cut using Aux 9033 (Z plane rotation). Only the first 45 degree segment was programmed. The actual program used is detailed as follows:

1. Tool 1001
2. X and Z offset RA
3. Tool 0
4. Z 0 RA
5. Tool 1
6. X 0 Y 1.875 RA Move to start position.

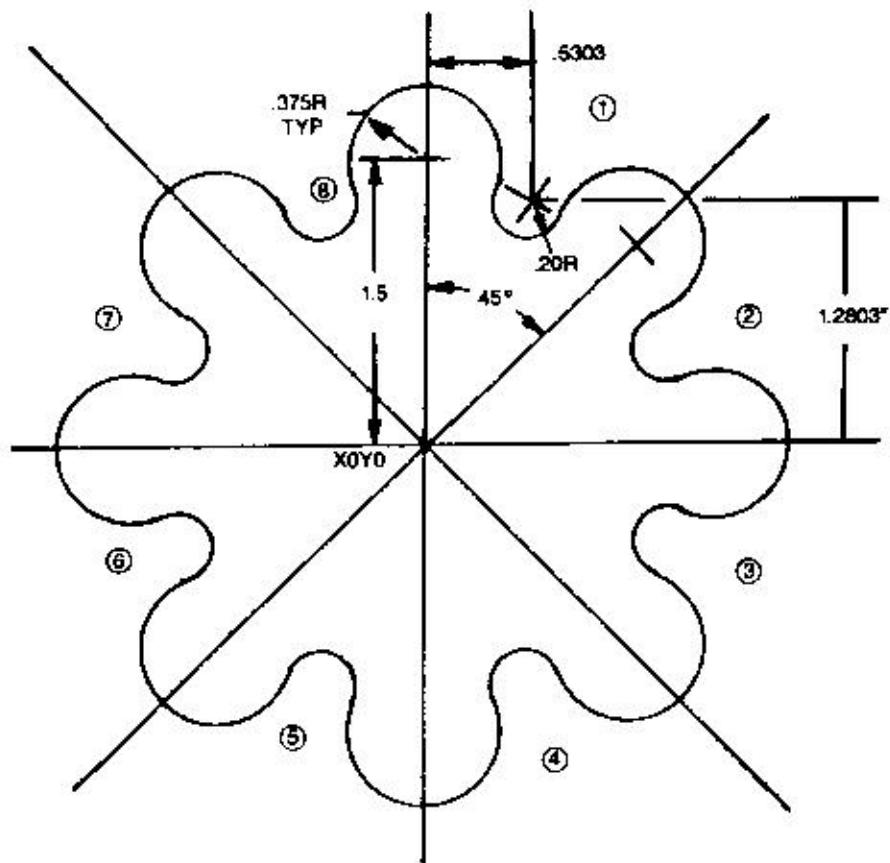


Figure 75

PLANE ROTATION

7. Z .1 RA	Move tool down
8. Feed 10	
9. Z-.1 FA	
10. Call 1	Call subroutine number one.
11. V60 -45.	Angle profile to be rotated.
12. Call 2	Call subroutine number two.
13. V60 -90.	Angle second rotation.
14. Call 2	
15. V60 -135.	Angle third rotation
16. Call 2	
17. V60 -180	Angle fourth rotation.
18. Call 2	
19. V60 -225.	Angle fifth rotation.
20. Call 2	
21. V60 -270	Angle sixth rotation.
22. Call 2	
23. V60 -315	Angle seventh rotation.
24. Call 2	
25. AUX 9030	Cancel rotation.
26. Z.1 FA	
27. Tool 0	
28. Z 0 RA	
29. END	
 (Subroutine 1 Part Profile)	
30. SUB	
31. Arc CW	
32. X 0 Y -.375 FI	

33. X .3542 Y -.4981 F1

34. Arc CW

35. Arc CCW

36. X .1889 Y -.0656 F1

37. X .3689 Y -.1528 F1

38. Arc CCW

39. Arc CW

40. X .3376 Y -.1634 F1

41. X .6027 Y .1017 F1

42. Arc CW

43. END

(Subroutine 2)

44. SUB 2

45. AUX 9033 Activates Z plane rotation

46. Call 1 Calls subroutine 1

47. Aux 9030 Cancels rotation

48. End

EXAMPLE TWO

Another example of rotation would be to rotate a pocket milling cycle. Zero must be set at bottom left corner of pocket. If this is not part zero a zero shift can be used.

The following program will rotate a 2" x 1" pocket 45 degrees.

```
1. Tool 1001
2. Z Offset RA
3. T 0
4. Z0 RA
5. X 0 Y 0 RC
6. T 1
7. Feed 10.
8. V40 .1
9. V41 .2
10. V42 .1
11. V43 -.25
12. V44 .25
13. V45 .175
14. V46 .25
15. V47 .010
16. V48 20.
17. V49 .25
18. V50 0
19. V60 45. Angle of Rotation
20. Aux 9033 Activate Rotation
21. G78
22. Aux 9030 Cancel Rotation
23. T 0
24. Z0 RA
25. END
```

SECTION 14
THE SERIES-M GRAPHICS
FEATURE

A. HOW TO USE GRAPHICS

To get into graphics the following steps must first be established:

1. Control must be in Program Check mode.
2. Control must have a program in memory
3. Tool 0 must be the active tool. Section A (right side) will indicate the active tool number.
4. Emergency Stop must be pulled out.
5. Press **DRAW**. This is the far left soft key. The summary screen will now appear on the CRT.

Set the necessary parameters on the summary screen by taking the following steps:

B. ZERO SET

In this step, you will determine where to set zero on the graphics screen. At start up, the zero's are always in default position, which is bottom left corner of screen. Please note graphics area is X 5.67 and Y 3.45.

- a) To get into Zero Set edit mode press 1.
- b) Press the **IN/MM** button to select inches or mm. Default is to inches.
- c) There is now a message on screen saying select 1-6. If 1-5 is selected a cross will appear on the screen at the selected point. If 6 is selected, the screen will change.

Line 1 will contain X axis dimension for your zero point. Line 2 will contain Y-axis dimension. Line 3 will contain Z axis dimension. Line 4 will have a message: **"SHOW POSITION ZERO ON SCREEN."** You can now put zero anywhere on or off screen.

Press 1 **DELETE**, this will erase the present X axis dimension. Enter the new value and press **ENTER**. A plus will move X to the right, minus will move it to the left (off the screen). If Y axis is to be changed, press 2 **DELETE** new value **ENTER**. Plus dimensions move up, minus moves down off the screen.

Now that X and Y axis 0 have been set press 4. This will show you where you have positioned it. In the case zero is off the screen a message will flash **"ZERO IS OUT OF GRAPHICS AREA."** When zero is placed in desired position, press **ENTER** to return to summary screen.

C. SCALING

Since the screen size is sometimes smaller or larger than the actual part, it may be necessary to scale the part dimension up or down in order to fit it into the screen.

- a) Press **2**. This will put you into the scale editing mode.
- b) If the present scale factor needs to be changed, press **DELETE**. Enter this new scale factor (for half-size, the scale factor would be **.5**) Check to see that the scale factor is correct.
- c) Press **ENTER** and the control will revert back to the normal summary screen.

NOTE:

Scaling does not affect X0, Y0, Z0 position on graphics screen.

D. EXECUTE BLOCKS

This section determines which block of the program will be graphically illustrated on the screen. The complete program or just a portion of it can be run.

- a) Press **3** to enter the execute block mode.
- b) If the starting block number is correct, press **NEXT**. (If it needs to be changed, press **DELETE** and enter the correct block number. Press **ENTER**, then press **NEXT**).
- c) After pressing **NEXT** the finish block will have the cursor flashing beside it. Use the same procedure described above to change the block number. Press **ENTER** again to get back to summary screen.

NOTE:

If complete program is being run and subroutines are being used, the end block of the main program is the finish block for graphics.

E. CUTTER COMPENSATION (ON OR OFF)

If cutter compensation is being used in the program, it can be drawn showing part profile (COMP OFF) or cutter path (COMP ON).

- a) Press **4** to enter Cutter Compensation Edit mode.
- b) The command selected will be flashing. At the bottom of the screen are command selections for ON or OFF. If the cutter compensation is off and needs to be on, press the softkey directly below the word **ON**, or vice-versa. If no change is necessary, press **Enter**. Either way, the control automatically reverts back to summary screen.

F. SPEED

This section determines the speed at which you want graphics to run.

- a) Press **S** to get into Speed Edit mode.
- b) Select number required and the control will revert back to the summary screen. If there are no changes, press **ENTER**.

G. RUNNING GRAPHICS

Now that the set-up information is correct, you can run the graphics. Ensure the **Emergency Stop** is pulled out, Tool 0 is active and press **START**. The graphics will start in a moment.

If the HOLD button is pressed, the graphics process will stop. Pressing **START** will restart it.

While graphics are running you can switch from AUTO to STEP by pressing **single step**. When in single step, **start** has to be pressed after each step (STEP will flash at the bottom right corner of the screen when it has completed an event) to go to the next step. This can be switched back and forth anytime running graphics.

Please note that the light lines on the graphics image are Rapid moves, heavy lines are Feed moves (2 axis) and very heavy lines are 3 axis Feed moves.

Planes may also be changed while graphics are running. Press the AUX key and (1) X Y; (2) X Z; (3) Y Z will appear across the bottom of the screen. The plane selection that is presently active will be flashing. To make a plane change, press the number desired and the screen will flip to that view.

SECTION 15
MISCELLANEOUS FEATURES
FOR THE SERIES-M

A. USING THE TAPE CASSETTE FEATURE

The tape cassette is used to store a program which has been entered in the Series "M" memory so that you can repeat the same program at a later time. Record programs only after it has been proven and when there will be no more changes.

To record a program on tape from Series-M memory:

1. Press **PROGRAM ENTER**.
2. Press **EMERGENCY STOP**.
3. Put a cassette tape in the recorder and close the spring-loaded flip-out door.
4. Press **RECORD**.

At this time, RECORD will flash in Area C, line 5. After a short time this message will change to SEARCH TAPE as tape rewinds. Next AREA C, line 5 will flash RECORDING which means it is recording Channel 1. When that is completed it will flash SEARCH TAPE again and rewind tape. It will then flash PLAYING at which time it is verifying Channel 1 and writing Channel 2. Finally Area C, line 5 will flash TAPE DONE. The program is now recorded and cassette can be removed from tape player and stored in a safe place.

There are 2 sides to each tape and a different program can be recorded on each side. Identify the tape by writing the part number or job number from the program sheet on cassette case.

To play a cassette back into Series-M memory:

1. Press **PROGRAM ENTER** and press **CLEAR 5** times to clear memory.
2. Press **EMERGENCY STOP**.
3. Put cassette tape in the recorder and close the spring loaded flip-out door.
4. Press **PLAY**.

At this time Area C, line 5 will flash PLAY. It will then flash SEARCH TAPE as it is rewinding. The next message is PLAYING as it is playing into memory. After this is complete the message will read TAPE DONE. Program is now in memory. Press **PROGRAM CHECK** to verify that the program has loaded in memory correctly.

If you have a program on a tape and record another program on the same side, the first program will be erased.

The control's memory will accept more than one program. This is done by playing the first tape as previously described and playing second tape into the memory in the normal way. The second program will push the first program further up in the memory. It is then possible to record both programs on one tape.

NOTE:

Do not put the tape close to a magnet. This will erase the programs stored on it.

The following is a list of error messages related to playing or recording tapes:

- | | |
|--------------------|---|
| 1. TAPE POWER | No cassette recorder power. The cassette unit does not have power to run the tape winding mechanism. |
| 2. NO TAPE | Cassette has no tape. Door not closed on cassette. |
| 3. RECORD TAB | Program cannot be recorded because safety tab is in wrong position. |
| 4. PLAY ERROR | The tape has not played correctly. Look at the program in Program Check and replay the tape if it is not correct. |
| 5. TAPE STOPPED | No program on the cassette you are trying to play or cassette has stopped. Door of cassette has been opened while playing or recording. |
| 6. RECORD ERROR | The program did not record on the tape, try again. |
| 7. NO PROGRAM | No program in memory to record. |
| 8. MEMORY EXCEEDED | Not enough space in memory for program that is trying to be played into memory. |

B. RS-232 AUX CODES

The following is a list of AUX codes to use RS-232, one from each of the six numbered sections would have to be entered into control in order to send from or receive into control. In order to enter these, the **Emergency Stop** button must be pulled out. Press **MANUAL**, **AUX** and the number required. **AUX 2700, 2701 or 2702** (whichever applies) would be entered last. After the first AUX code is entered, press the (AUX) button twice before entering second number.

- | | |
|-------------|--|
| 1. AUX 2700 | Write to RS-232C device in RS-274 format (G code). |
| AUX 2701 | Read from RS-232C device in RS-276 format |
| AUX 2702 | Write to RS-232C device in Anilam format |
| 2. AUX 2754 | Use EIA character set |
| AUX 2758 | Use ASCII |

- | | | |
|----|-----------|------------------------------------|
| 3. | AUX 2765 | Set 5 bits per character |
| | AUX 2766 | Set 6 bits per character |
| | -AUX 2767 | Set 7 bits per character |
| | AUX 2768 | Set 8 bits per character |
| 4. | AUX 2770 | Set to no parity |
| | AUX 2771 | Set to odd parity |
| | -AUX 2772 | Set to even parity |
| 5. | AUX 2780 | Set baud rate to 110 bits/second |
| | AUX 2781 | Set baud rate to 150 bits/second |
| | AUX 2782 | Set baud rate to 300 bits/second |
| | AUX 2783 | Set baud rate to 600 bits/second |
| | AUX 2784 | Set baud rate to 1200 bits/second |
| | AUX 2785 | Set baud rate to 1800 bits/second |
| | AUX 2786 | Set baud rate to 2400 bits/second |
| | AUX 2787 | Set baud rate to 4800 bits/second |
| | -AUX 2788 | Set baud rate to 5600 bits/second |
| | AUX 2789 | Set baud rate to 19200 bits/second |

Any speed higher than 1200 bits/second will transfer data in bursts (must use handshake).

- | | | |
|----|-----------|-------------------------------------|
| 6. | AUX 2790 | Set no handshake |
| | -AUX 2791 | Set software handshake (X ON X OFF) |
| | AUX 2792 | Set hardware handshake (DTR, DSR) |

The Crusader Series-M has the ability to run programs that exceed the unit's memory capability. This is called Continuous Download. The program will be transmitted from an RS-232C device to the Crusader Series-M. First, certain parameters must be set using AUX codes manually programmed into the Series-M. The character set (EIA or ASCII), bits per character, parity, bits per second (baud rate) and handshake must match that of the transmitting device.

TO SET THE RS-232-C PARAMETERS FOR TRANSFER:

Put the Series-M in manual mode. Press AUX 2758 (or 2754), then START to select ASCII (RS-358) or EIA (RS-244-A) character set.

Press AUX 2768 and 2770, then START to select 8 bits per character without parity...or, press AUX 2767 and 2771 (or 2772), then START to select 7 bits per character with odd or even parity.

Press AUX 2780 (or 2781, 2782, 2783, 2784, 2785, 2786) then START to select 110 (or 150, 300, 600, 1200, 1800, 2400) bits per second, respectively. The baud rate must not exceed 2400 bits per second.

Press AUX 2791 then START to select software or hardware handshake. Put AUX 2711 into the user memory in block one, enter the AUTO mode and press 1, SEARCH, START.

The Series-M will wait for the transmitting computer to send its program. The program must be divided into sections that must fit in the available memory of the Series-M. Each section must start with a percent (%) sign and be terminated the same way. Programs must not contain any CALL, DO, SUBR or END

statements. When the transfer of a section is completed, the Series-M will put the transmitting computer on hold and will start executing the section of program just loaded from the RS-232-C link.

When the Series-M reaches the end of a section, it will request and accept more data from the RS-232-C device. It is important to understand that the control will pause while accepting more information. In this way, retraction of the quill between program sections can be programmed.

When the end of the program is encountered, execution stops and the RS-232-C link is interrupted.

C. SERIES-M ERROR CODE MESSAGES

RS-232 MESSAGES:

INVALID 2700	Wrong code entered into control.
RS-232 ABORT	If emergency stop is pressed during transmission, this message will appear.
RS-232 OVR11	Control memory exceeded.
RS-232 READ	Playing into control.
RS-232 WRITE	Playing out of control.
BUFFER OVRLL	Transmitting data too fast.

ERROR MESSAGES:

Excess Digit	Too many numbers or letters entered.
Circle error	Circle formats incorrect.
No Intersect	When using cutter compensation cutter cannot get to next point.
Limit Switch	One axis is on a limit switch.
Z not rapid	Z command too fast.
Excess Accel	Acceleration too high.
Neg Velocity	Negative velocity. Opposing distance and velocity commands.
Lag Error	Following error increasing too fast.
Excess Vel	Excess velocity. Velocity command too fast.
No cut comp	Cutter compensation not allowed with present canned cycle programmed.
Tool Def Err	No tool diameter in offset event

Zero feed	No feed programmed.
Neg Sqrt	Negative square root.
Pos Drill	Positive drill. Z dimension in wrong direction.

GENERAL MESSAGES:

Complete	Control has been turned on correctly.
Failed	Control has not come up correctly.
Drift Register	Drift registers being checked.
Error Register	Error registers being checked.
Emergency Stop	Emergency stop is depressed.
Dwell Forever	Dwell forever. Either there is a dwell programmed or there is an error in a canned cycle.
Tool change	Tool change programmed.
HOLD	Hold button has been pressed.
X + Limit	X + software limit.
X - Limit	X - software limit.
Y + Limit	Y + software limit.
Z + Limit	Z + software limit.
Z - Limit	Z - software limit.

E. DIAGNOSTICS:

Before entering diagnostics, be sure to tape any programs on a cassette. Some of these tests will clear the memory and you will lose any programming information contained in the memory.

To enter diagnostics, the control must be in the PROGRAM ENTER mode. Press the **TEST** soft key. Area D on the screen will now display a series of letters and numbers that are for software identification. If **TEST** is pressed again, the control will go into diagnostics. If **END** is pressed, the control will revert back to the normal programming screen in Manual mode.

When control is put into diagnostics mode, there will be a list of tests that can be selected and performed.

The first screen will check the 68000 Processor (503 and 500 boards) and the following lists of tests will appear:

- | | |
|---------------------|---|
| 0. 8083 DIAGNOSTICS | Changes screen and allows 8085 processor to be checked (513 board). |
| 1. EPROM CHECKSUMS | |
| 2. SYSTEM RAM TEST | |
| 3. DISPLAY RAM TEST | |
| 4. USER RAM TEST | |
| 5. DUART TEST | |
| 6. 6840 TIMER TEST | |
| 7. KEYBOARD TEST | |
| 8. CRT PATTERN | There are two patterns. Press any key for second pattern. |
| 9. CONTINUOUS TEST | This does all the above tests in sequence. |

If "D" is pressed, the control will change screens and the 8085 processor can now be checked. The following lists of test will appear on the screen:

0. EXIT TO 68000 DIAGNOSTICS
1. EPROM CHECKSUMS
2. SYSTEM RAM TEST
3. I.D. REQUEST
4. USER RAM TEST
5. APU TEST
6. COUNTER TEST
7. CONTINUOUS TEST

If there is an error while running any of these test, then a failure message will be displayed on the screen. To return to selection screen while in a test (or when tests are completed), press END.

To return control to regular operating modes, first the control must be in 68000 DIAGNOSTICS. Then press the Softkey on the far right. COMPLETE will flash in Area C, line 5.

If further help with diagnostics is required, call your nearest Anilam representative; or call the Anilam Service Hot-Line in Miami at:

1-800-327-6340

SECTION 16:
ADDITIONAL DEMONSTRATION EXAMPLES

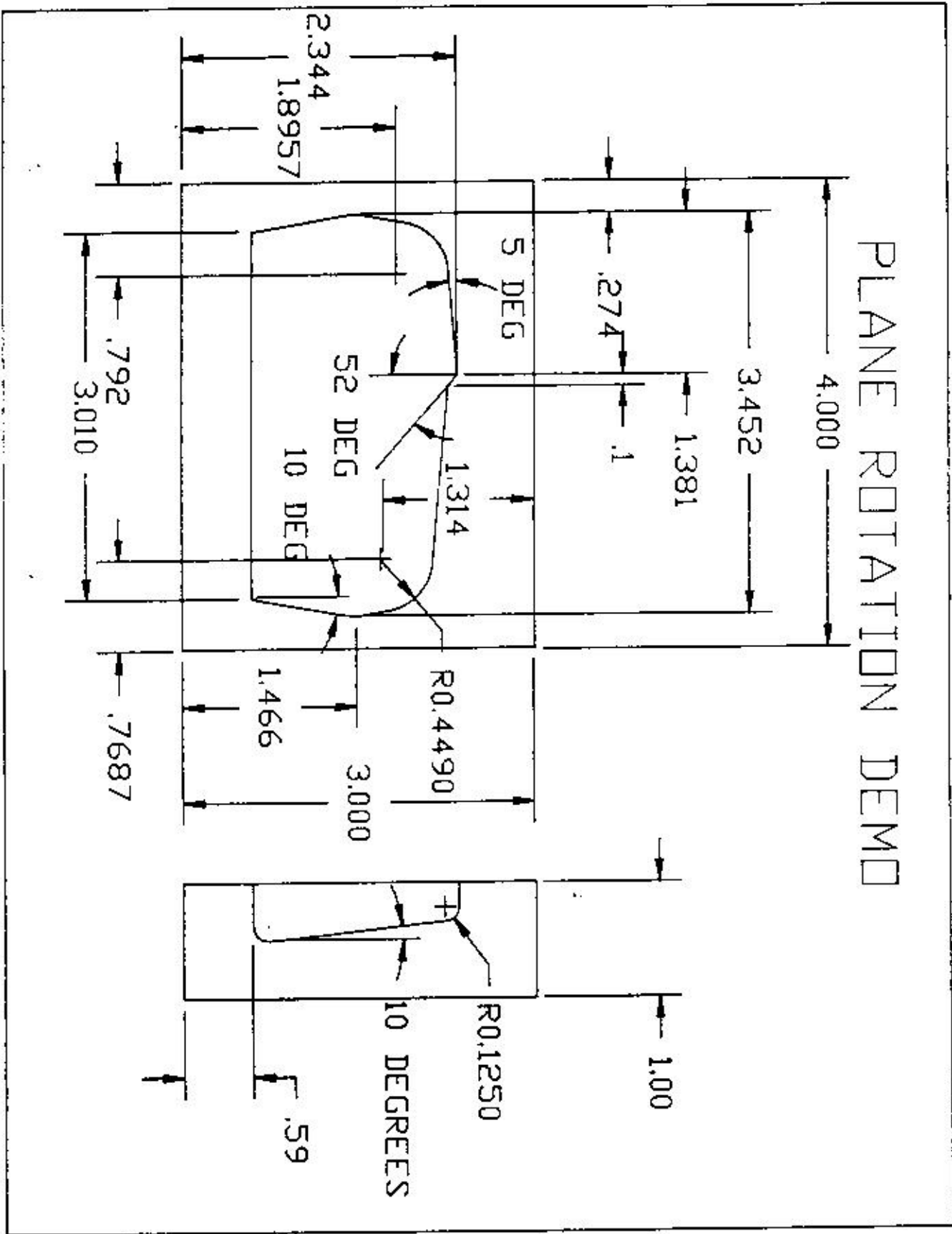


Figure 76

A. PLANE ROTATION DEMO PROGRAM (Figure 76):

Tool #1 1/4in Endmill
Tool #2 - #5 1/4in Ball endmill

```

1.  TOOL 1001
2.  X .25                      Z-1.0000 RA
3.  TOOL 1002
4.  X .6250                    Z-1.0000 RA
5.  TOOL 1003
6.  X .5000                    Z-1.0000 RA
7.  TOOL 1004
8.  X .3750                    Z-1.0000 RA
9.  TOOL 1005
10. X .2500                    Z-1.0000 RA
11. TOOL 0
12.                              Z .0000 RA
13. X-1.375      Y.7000 RA
14. TOOL 1
15. FEED 10.
16. V40 .1
17. V41 2.7500
18. V42 1.3000
19. V43 -.3500
20. V44 .1250
21. V45 .1750
22. V46 .1750
23. V47 .0000
24. V48 .0000
25. V50 1.0000
26. G78
27. TOOL 0
28. Z .0 RA
29. X-3.0000      Y1.0000      RA
30. TOOL 5
31. AUX 1101
32. X-1.5700      Y .6500      Z-1.0000      RA
33. X .0000      Y .0000      RA
34. Z .1000      RA
35. V40 .1000
36. V41 3.1420
37. V42 1.3650
38. V43 -.6100
39. V44 .4490
40. V45 .0625
41. V46 .6100
42. V47 .0000
43. V48 .0000
44. V50 1.0000
45. V60 10.0000
46. AUX 9031
47. G78
48. AUX 9030

```

Tools 2-5 are the same tool, the diameter has been changed to allow same tool to be used for rough and finish cuts using cutter comp.

Regular rectangular pocket cycle

Zero shift to move zero to bottom left corner of pocket. This is necessary in order to rotate pocket.

Rotated rectangular pocket cycle.
V60 = Angle to be rotated
AUX 9031 Activate rotation around X

AUX 9030 Deactivate rotation


```

49. TOOL 0
50. Z .0000 RA
51. TOOL 2
52. X .0000 1.2500 RA
53. Z .1000 RA
54. Z .0000 FA
55. DO 5
56. Z -.1030 FI
57. CALL 1
58. END
59. Z .1000 FA
60. TOOL 0
61. Z .0000 RA
62. TOOL 3
63. X .0000 Y 1.2500 RA
64. Z .0000 RA
65. DO 5
66. Z -.1030 FI
67. CALL 1
68. END
69. Z .1000 RA
70. TOOL 0
71. Z .0000 RA
72. TOOL 4
73. X .0000 Y 1.2500 RA
74. Z .0000 RA
75. DO 5
76. Z -.1030 FI
77. CALL 1
78. END
79. Z .1000 FA
80. TOOL 0
81. TOOL 5
82. X .0000 Y1.2500 RA
83. Z .0000 RA
84. DO 5
85. Z -.1030 FI
86. CALL 1
87. END
88. Z .1000 RA
89. TOOL 0
90. Z .0000 RA
91. X-5.0000 Y1.0000
92. END
93. SUB 1
94. V60 10.0000
95. AUX 9031
96. G41
97. Y .5810 FA
98. X 1.5715 FA
99. X 1.5715 Y1.4700 FA
100. X 1.6735 Y1.7710

```

Rotate following
moves around X axis

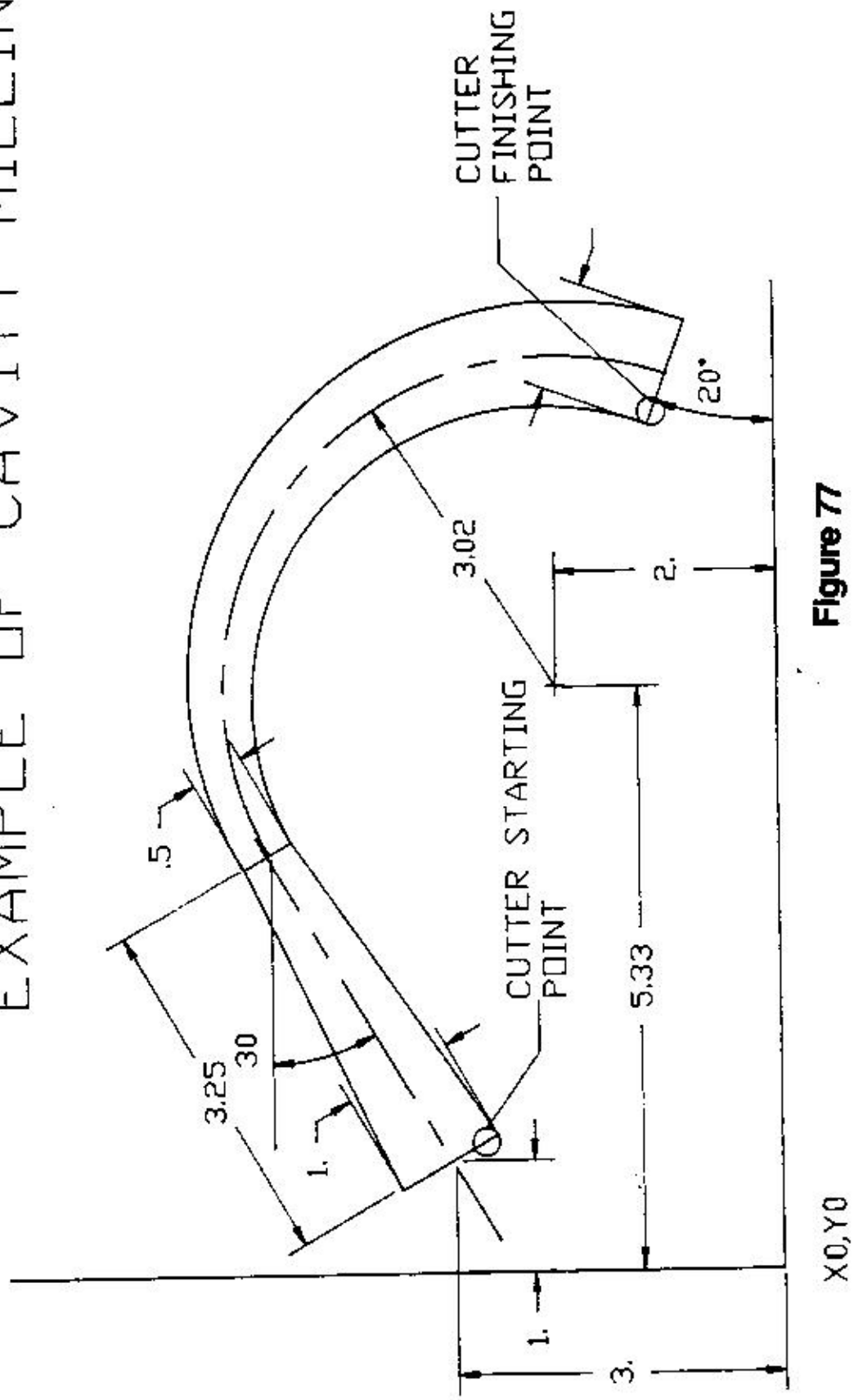
FA

102.	ARC CCW		
103.	X 1.2313	Y1.6930	FA
104.	X 1.2704	Y2.1410	FA
105.	ARC		
106.	X -.2450	Y2.2800	FA
107.	X -.3450	Y2.3610	FA
108.	X-1.2472	Y2.2800	FA
109.	ARC CCW		
110.	X-1.2080	Y1.8220	FA
111.	X-1.6502	Y1.8957	FA
112.	ARC		
113.	X-1.7260	Y1.4700	FA
114.	X-1.5715	Y .5810	FA
115.	X .5000		FA
116.	G40		
117.	Y1.2500		FA
118.	AUX 9030		
119.	X .0000	Y1.2500	FA
120.	END		

NOTE:
All Y axis dimensions
have been adjusted to
allow for 10 degree
angle.

Cancel rotation

EXAMPLE OF CAVITY MILLING



B. CAVITY AND ELBOW CAVITY MILLING PROGRAM (Figure 77):

Tool #1: .25 ball endmill
 XO and YO: Bottom left corner

1.	TOOL 1001			
2.	X .2500	Z1.0000	RA	
3.	TOOL 0			
4.	Z.0000	RA		
5.	TOOL 1			
6.	X 1.0000	Y3.0000	RA	Zero shift moves
7.	AUX 1101			to center of arc
8.	X 5.3300	Y2.0000	Z-1.0000 RA	elbow cavity.
9.	V14 .3750			Rapid polar move to
10.	V15 -60.0000			cutter start point.
11.	G10			
12.	Z .2250	RA		
13.	FEED 10.			
14.	Z .0000	FA		
15.	V60 30.000	Angle of cavity		
16.	V62 3.2500	Length of cavity		
17.	V63 7.0000	Number of cycles		
18.	V64 .3750	Initial radius		
19.	V65 .1250	Finish radius		
20.	V66 .0000	Concave		
21.	G48	Activate cycle		
22.	V62 -20.0000	Finish angle		
23.	V63 14.0000	Number of cycles		
24.	V64 .1250	Initial radius		
25.	V65 .3750	Finish radius		
26.	V66 .0000	Concave		
27.	V79 - 1.0000	Clockwise direction		
28.	G49	Activate cycle		
29.	Z .2250 RA			
30.	AUX 1101			
31.	X .0000	Y .0000	Z .0000 RA	
32.	TOOL 0			
33.	Z .0000 RA			
34.	X-3.0000	Y .0000 RA		
35.	END			

Technical drawing of a mechanical part with dimensions and a coordinate system. The drawing shows a profile with the following dimensions and features:

- Top horizontal edge: 1.500
- Top-left corner radius: R0.2945
- Top-left corner offset: .2945
- Top-right corner radius: R0.2945
- Right vertical edge: 7.906
- Right vertical edge offset: 5.2513
- Right vertical edge offset: 1.918
- Right vertical edge offset: 3.112
- Right vertical edge offset: .9432
- Right vertical edge offset: R1.4487
- Right vertical edge offset: R0.5835
- Right vertical edge offset: 1.2533
- Right vertical edge offset: 106°
- Right vertical edge offset: 24.7181 RAD CENTER X23.7952 Y7.4957
- Right vertical edge offset: R1.9934
- Right vertical edge offset: X0Y0

Figure 78

C. MOLD ROTATION EXAMPLE (Figure 78):

TOOL #1: .5 Diameter Ball Endmill
X0 and Y0: Top center of part
Z0: Set .25 below top surface (radius of cutter)

```
1.  TOOL 1001
2.  X .5000 Z-1.0000 RA
3.  TOOL 0
4.  Z .0000 RA
5.  X-3.0000 Y2.0000 RA
6.  TOOL 1
7.  FEED 20
8.  X -.0000 Y-.2500 RA
9.  Z .3500 RA
10. Z .0000 F
11. V61 .0000 Starting angle
12. V62 180.0000 Finishing angle
13. V63 10.0000 Number of cycles
14. V70 1.0000 Subroutine number 1
15. V71 2.0000 Subroutine number 2
16. G46
17. Z .3500 FA
18. TOOL 0
19. Z-.0000 RA
20. X-3.0000 Y2.0000 RA
21. END
```

V70 subroutine #1

```
30. SUB 1 FI
31. X 1.5000
32. ARC CW
33. X .0000 Y-.0445 FI
34. X .0314 Y-.0761 FI
35. ARC
36. ARC CCW
37. X 1.5804 Y-1.5919 FI
38. X .4613 Y-2.5211 FI
39. ARC
40. ARC CCW
41. X22.7251 Y10.3429 FI
42. X .9200 Y-1.8202 FI
43. ARC
44. ARC CW
45. X-1.0469 Y-.5539 FI
46. X .1050 Y-.9121 FI
47. ARC
48. X-.5221 Y-1.8343 FI
49. ARC CW
50. X-.3207 Y .0913 FI
```

51.	X-.3207	Y -.2422			
52.	ARCI				
53.	X-1.2533	FI			
54.	AUX 9030				Deactivate rotation
55.	X .0000	Y-7.6560	Z .0000	FA	Reposition in absolute
56.	END				
60.	SUB 2				V71 subroutine #2
61.	X 1.2533	FI			
62.	ARC CCW				
63.	X .0000	Y .3335	FI		
64.	X .3207	Y .2422	FI		
65.	ARC				
66.	X .5221	Y1 .8343	FI		
67.	ARC CCW				
68.	X-1.1529	Y .3282	FI		
69.	X- .1060	Y .9121	FI		
70.	ARC				
71.	ARC CW				
72.	X21.8051	Y12.1631	FI		
73.	X- .9200	Y 1.8202	FI		
74.	ARC				
75.	ARC CW				
76.	X 2.0417	Y.9292	FI		
77.	X .4613	Y 2.5211	FI		
78.	ARC				
79.	ARC CCW				
80.	X- .0314	Y .0316	FI		
81.	X- .0294	Y .076	FI		
82.	ARC				
83.	X-1.5020				
84.	AUX 9030				Deactivate rotation
85.	X .0000	Y -.2500	Z .0000	FI	Reposition in absolute
86.	END				

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # _____ PROGRAMMER _____ DATE _____

TOOL LIST

OFFSETS

SETUP INSTRUCTIONS

EVENT NO.				RAPID / FEED	INCR / ABS	EVENT NO.				RAPID / FEED	INCR / ABS
1						1					
2						2					
3						3					
4						4					
5						5					
6						6					
7						7					
8						8					
9						9					
0						0					
1						1					
2						2					
3						3					
4						4					
5						5					
6						6					
7						7					
8						8					
9						9					
0						0					

