MILL*PWR* Setup Access Code

An access code must be entered before the installation setup parameters can be accessed or changed.

IMPORTANT
The access code is **8891**.

Refer to Section 7, **Setup**.

IMPORTANT
Supervisors may wish to remove this page from the **MILL***PWR* Operation Manual after initially setting up the installation setup parameters. Keep it in a safe place for future use.

TABLE OF CONTENTS

MILL*PWR*®

ACU-RITE®

⚠

This symbol alerts you to important information
concerning the operation of your **MILL*PWR*** system.

MILL*PWR* System Specifications

Characteristic	Specification
Operating conditions	0° to 40° C (32° to 104° F) 25% to 85% relative humidity (non-condensing)
Storage conditions	-20° to 60° C (-4° to 140° F) 25% to 95% relative humidity (non-condensing)
Input requirements: Voltage Frequency Current	 115V~ (±20%), single phase 47 - 63 Hz 8.5A rms nom., 18A rms peak—inrush
Fuse	15A/250V resettable circuit breaker

Encoder input	Position signals, channels A & B
	        TTL square wave signal in quadrature
	        (90° nominal phase relationship)
	        Maximum input rate: 50 kHz
	
	Reference signal: TTL square wave

Edge finder input	Compatible with ACU-RITE® Electronic Edge Finder
FCC compliance	Class A

INTRODUCTION

System Overview

Operator
Console

ENC 150
Precision
Glass
Scales

Motor
Drive
Assemblies

Front View of Operator Console

Power Indicator

Hard Key Milling Functions

10" Flat-panel, Color, LCD Display

Keypad

Table Stop

Softkeys

Floppy Disk Drive

Rear View of Operator Console

(Heat Sink)

TEST   PENDANT   TOUCH PROBE

W AXIS ENCODER   Z AXIS ENCODER

SERIAL PORT

EDGE FINDER

Power Switch

AMI

Z AXIS SERVO   Y AXIS SERVO   X AXIS SERVO

SERVO POWER

Ground Post   Power Input
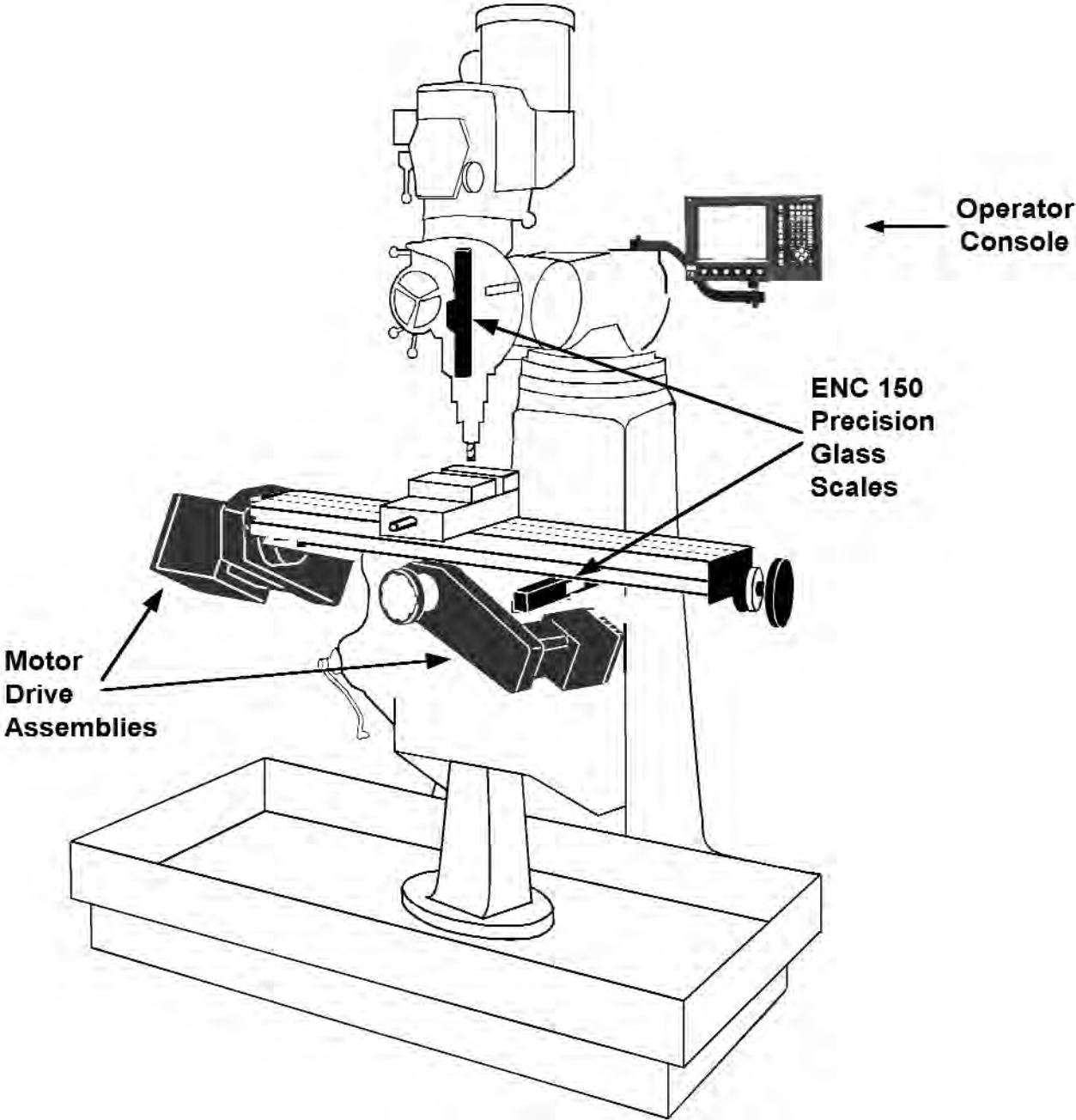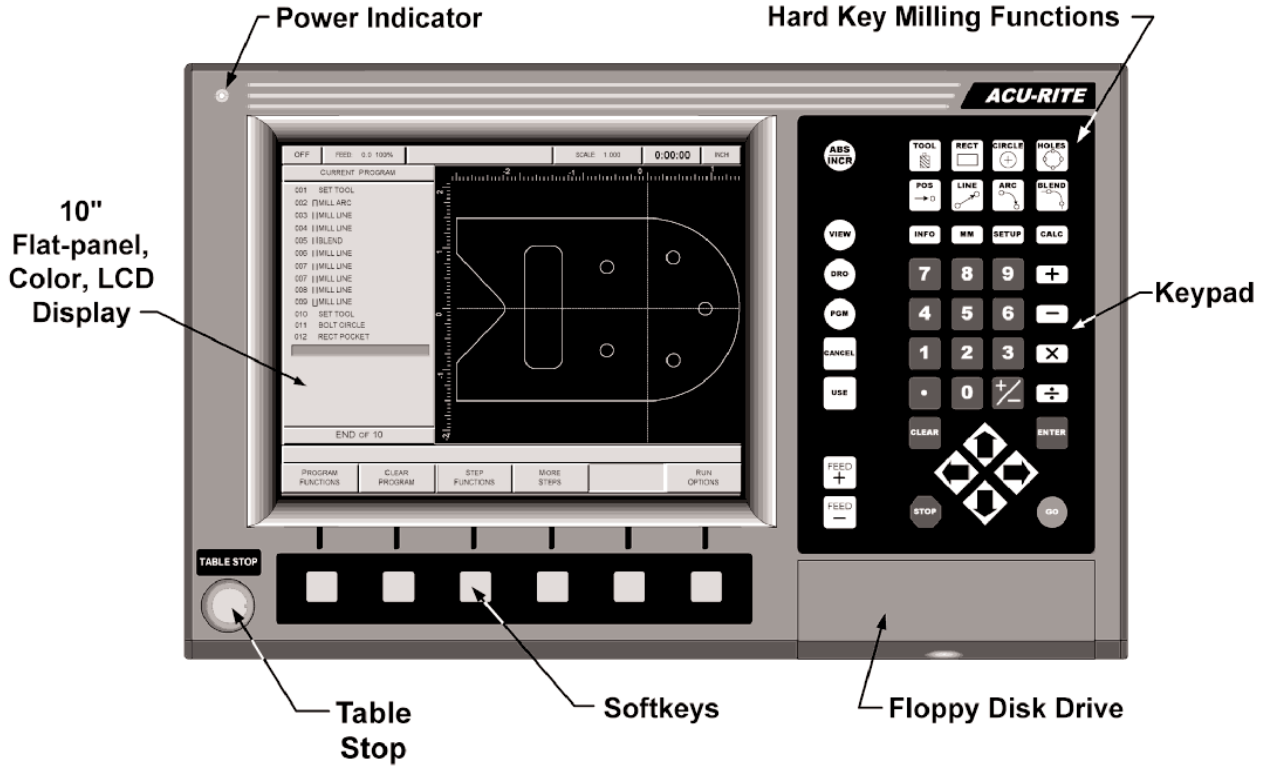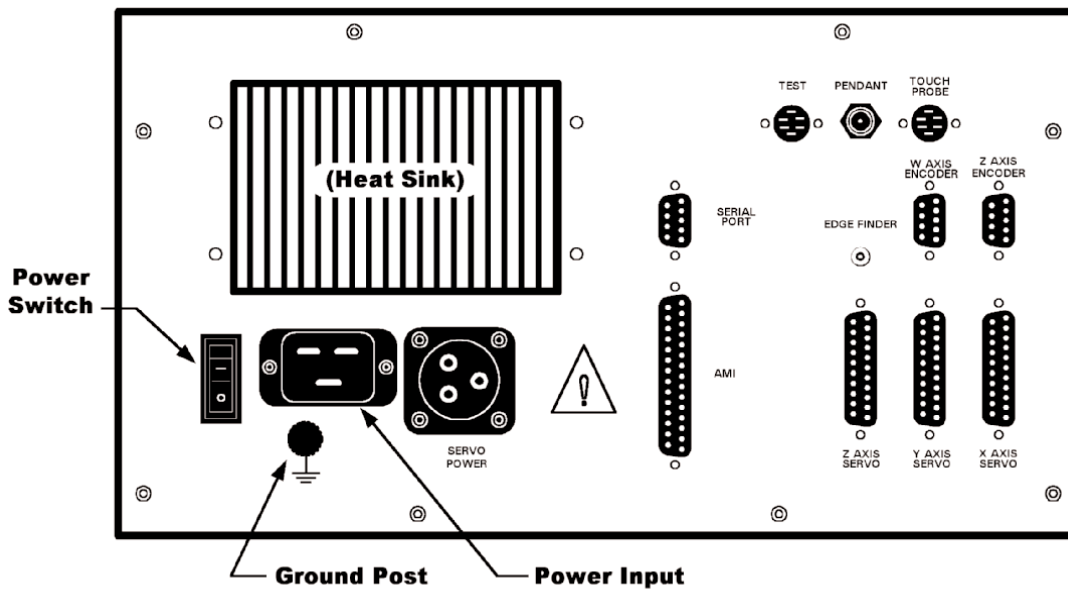
Keypad Layout

Main Function Keys

ABS/INCR   Switch from absolute to incremental (or vice versa) in the DRO display and milling function numeric fields.

VIEW   Manipulate a part graphic.

DRO   Display the digital readout.

PGM   Display the program screen.

CANCEL   Exit from a milling function.

USE   Add a step to the program once you have completed an entry form.

Cursor and Motion Control
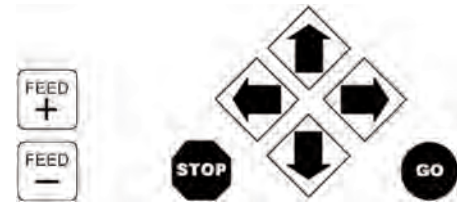
GO   Start a program.

STOP   Press this key once to pause the program, twice to exit.

FEED+
FEED-   Increase or decrease the feed rate.

ARROWS   These keys enable you to position the table or move the cursor.

Special Function Keys

INFO   Access information about any **MILL*PWR*** function.

MM   Switch from inches to millimeters or vice versa.

SETUP   Add to the tool library, set feed rates, change display options and define other system parameters.

CALC   Perform standard (+, -, x, ÷), trigonometry, geometry and RPM calculations.

Milling Functions

TOOL   RECT   CIRCLE   HOLES
POS   LINE   ARC   BLEND
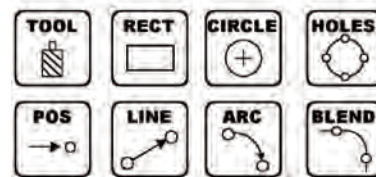
Use these keys to create a program. All but **BLEND** may also be used as one-time milling functions from the DRO display.

Numeric Keypad and Calculator

Enter program data and perform math calculations. Press the **CLEAR** key to delete information from a data field. Press the **ENTER** key to accept the information you have entered.

7 8 9 +
4 5 6 −
1 2 3 ×
. 0 +/− ÷
CLEAR   ENTER
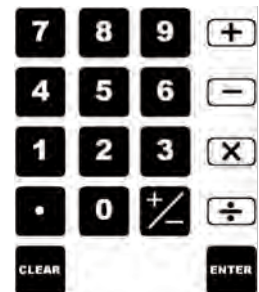
Screen Layout

The **MILL*PWR*** display screen is divided into four sections.
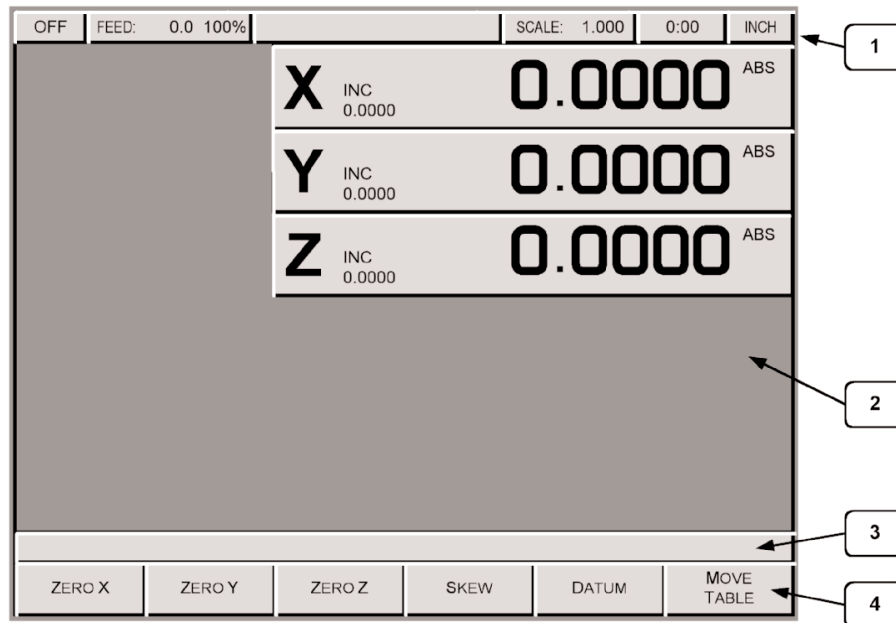
1 ▸ **Status bar** - displays the servo motor status (ON/OFF), feed rate, current tool, scale, job clock, and the current display setting (inches or millimeters).

2 ▸ **Information area** - displays information about the job being performed.

- **Readout (DRO)** - used as a digital readout, the display will show the current position for each axis.

- **Program (PGM)** - when programming, a list of program steps (milling functions) and part-view graphics will be displayed.

- **Calculator (CALC)** - a TRIG, R.P.M., and geometry calculator are available to assist with calculating missing information. They can be used as stand-alone calculators or, while programming, the calculations can easily be returned to the program.

3 ▸ **Message line** - operator prompts and messages will appear here.

4 ▸ **Softkeys** - variable milling functions appear here; functions are selected by pressing the softkey directly below each category. When a key appears pressed, it is selected. When it appears "raised" it is not selected.

Table Stop Button

The large red button located in the lower left corner on the front of the **MILL*PWR*** operator console is the **TABLE STOP**. In the event of a malfunction or programming error, press the **TABLE STOP** button to turn off the servo motors. *Turning off the servo motors will immediately stop all table movement.*

WARNING
Pressing the **TABLE STOP** button will **NOT** stop the rotation of the cutting tool unless the machine has been configured to do so. In the event of an emergency, if the machine has not been wired to stop the rotation of the cutting tool, be prepared to raise the tool in addition to pressing the **TABLE STOP** button.

Conventions

Axis Conventions

Count Direction

When programming a part using **MILL*PWR***, table movement and tool movement are determined by the use of positive or negative numbers. **MILL*PWR*** has been factory set with the following positive and negative count directions for the X, Y and Z-axes:

X-axis: The table will move to the left and the tool will move to the right for a positive count direction.

Y-axis: The table will move toward you while the tool moves away from you for a positive count direction.

Z-axis: The quill will move up (away from the table surface) for a positive count direction.

INTRODUCTION
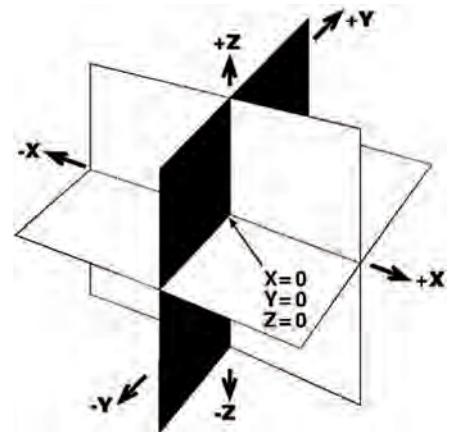
Cartesian Coordinates

A cartesian coordinate is a position that can be measured from the X- and Y-axes.
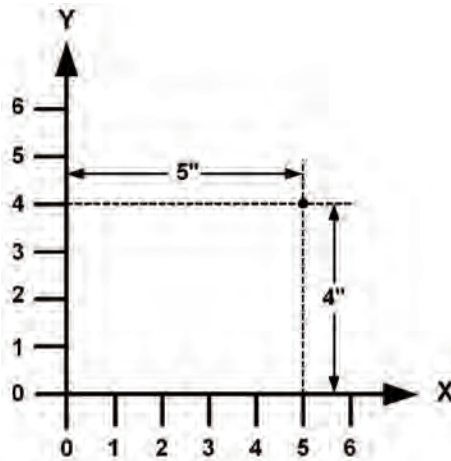
Polar Coordinates

A polar coordinate is a position that is defined by an angle and a radius.
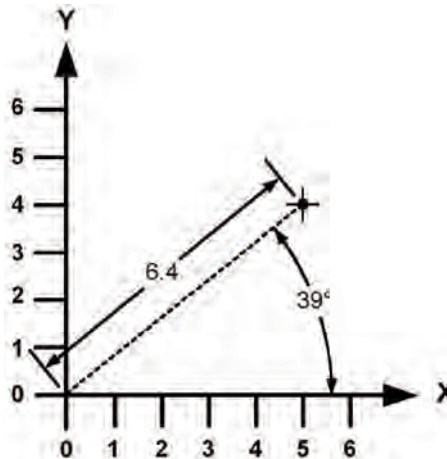
Absolute and Incremental Dimensions

Dimensions that you enter from a print are either absolute or incremental.

Absolute dimensions are measured from datum (also known as workpiece zero). *Incremental dimensions* are measured from one point to another.

Holes A and B are dimensioned using absolute values. Hole C is dimensioned incrementally from Hole A.
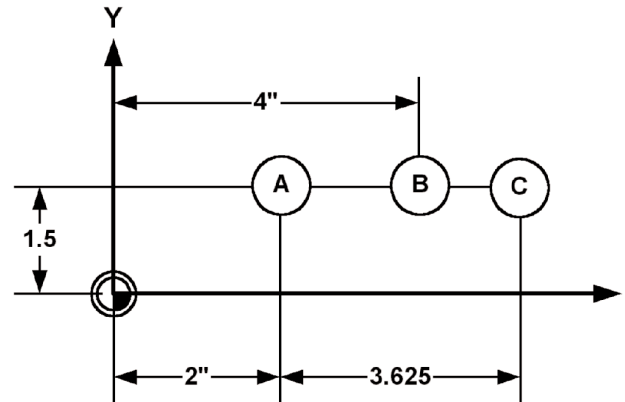
When entering these dimensions, we would say:

Hole A:	Position/Drill	002
	X 2.0000 ABS	
	Y 1.5000 ABS	

Hole B:	Position/Drill	003
	X 4.0000 ABS	
	Y 1.5000 ABS	

Hole C: Position/Drill   004
                X 3.6250 INC 002 - This indicates that the X position will increment
                Y 1.5000 ABS   from the X value in Step 2 (Hole A) above.

Note: Both absolute and incremental dimensions may be used to define a position, as shown with Hole C.

It's often easier to describe a location in terms of incremental dimensions rather than calculate its absolute coordinates.

Saving, Backing Up, and Creating Directories for Programs

If you are creating a long program, dont wait until the end to save your work.

Frequent saving reduces the risk of losing your work due to a power interruption.

When you create programs with **MILL*PWR***, you can save them in any of three places—within **MILL*PWR***'s internal memory, on a 3$^1/_2$" floppy disk, or on your PC. Saving your work means it will not be lost if **MILL*PWR*** is powered down or if there is a power interruption. As you save and back up your programs, you can neatly organize them in any of the following three main directories ("MILLPWR," "A:" and "REMTSTOR") or in personalized subdirectories that you can create.

MILL*PWR* is also equipped with a backup feature that enables you to make duplicate copies of your saved programs. We recommend that you back up your programs regularly to avoid accidental loss or other problems that may prevent you from recovering your original programs. Backing up programs takes only a few moments—and will save you valuable time if a problem does occur.

Note: Before you save or back up programs onto your PC, refer to **Remote Storage** and **Setup** for instructions.

For more details about how to save programs, back up files and create directories, refer to **Programming**.

DRO

Start Up

Power Up

Turn the power switch (located on the rear of the operator console) to "I." The Power Indicator (located in the upper left corner of the operator console) will light up.

```
OFF │ FEED:    0.0 100% │                          │ SCALE: 1.0000 │ 0:00:00 │ INCH

        X  INC              0.0000  ABS
           0.0000

        Y  INC              0.0000  ABS
           0.0000

        Z  INC              0.0000  ABS
           0.0000

           ⚠ CAUTION

           NEED AT LEAST 2 INCHES
             TRAVEL IN ALL AXES

             WHEN TOOL IS CLEAR
            PRESS FIND HOME OR

                  CANCEL

POWER WAS OFF.  FIND HOME TO RECALL PREVIOUS DATUM.
                                          │ FIND │         │ CANCEL │
                                          │ HOME │         │        │
```
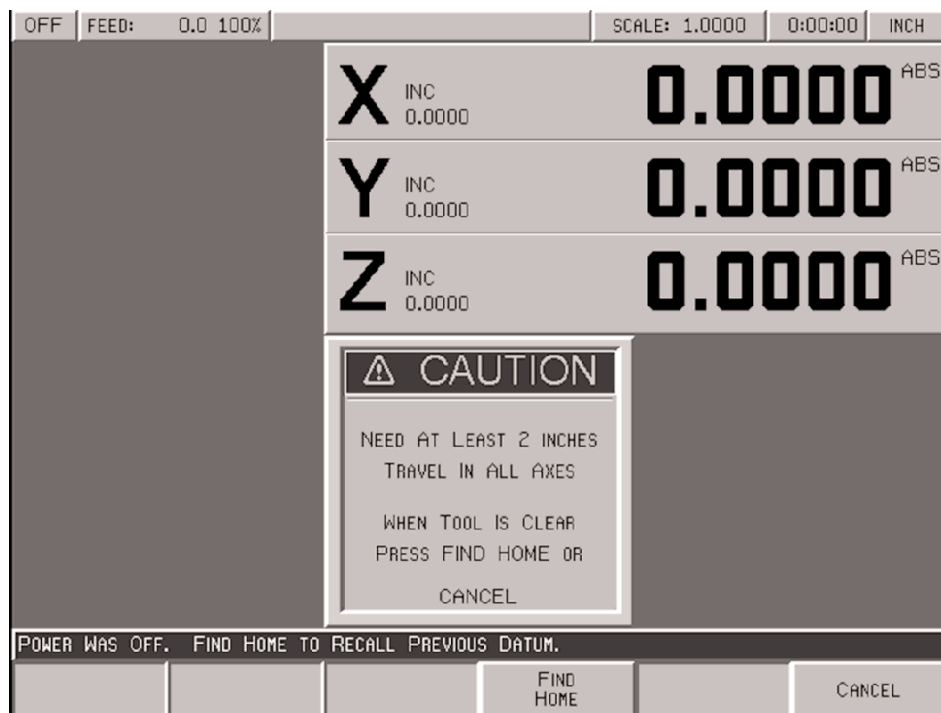
Screen Saver

Anytime the system is inactive for approximately 20 minutes, the LCD display will shut off, and a blank screen will appear. This screen saver function is designed to prolong the life of the operator console display. If the screen is blank, check that the Power Indicator light is illuminated. Press any key on the operator console or move the table and the display will reappear. If the Power Indicator light is not illuminated, then power to the **MILL*PWR*** system has been interrupted.

Finding Home

⚠   If you don't find home before moving the table, you will risk exceeding the table's travel limits and damaging the milling machine, **MILL*PWR*** or both.

You must find home before you run a program. To find home immediately after startup, press the **FIND HOME** softkey. Otherwise, press the **DATUM** softkey, then the **FIND HOME** softkey. The table will automatically move a few inches along the X- and Y-axes to find home.

Datum is a term used to describe "workpiece zero" or "absolute zero."

After you power up and find home, you can quickly move to datum by pressing the POS key, followed by the GO key.

When finding home, **MILL*PWR*** will use ACU-RITE's advanced Position-Trac™ technology. Position-Trac works by using a very precise distance-encrypted reference mark line pattern that's been placed onto each ACU-RITE precision glass scale included with the **MILL*PWR*** system. Proprietary software decodes the line pattern which then allows you to accurately find home and reestablish workpiece zero from any position.

With Position-Trac, there is no need to leave the system powered up when it is not being used. You will be able to easily, quickly and accurately reestablish workpiece zero immediately after you restart **MILL*PWR***.

After home has been found, the tool's position (relative to your most recent datum) will be displayed.
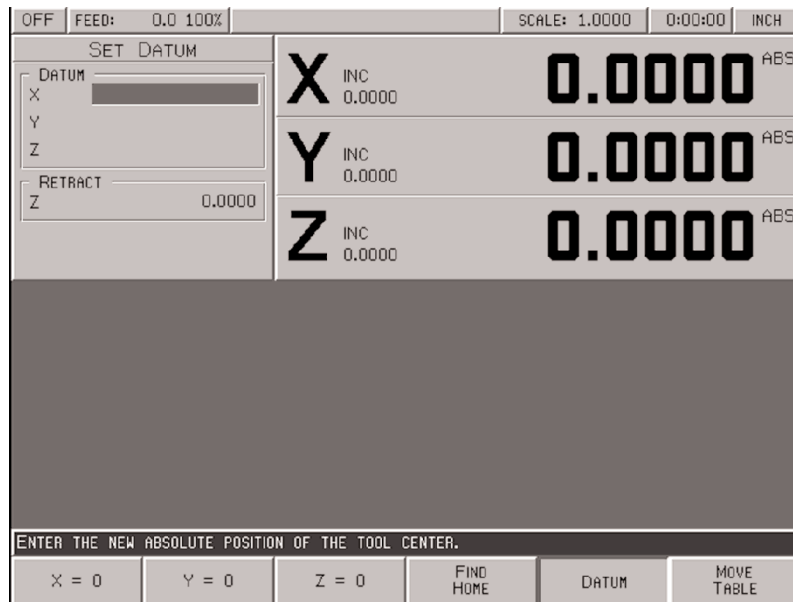
DRO Functions

The digital readout (DRO) display shows you the current tool position. While operating in the DRO mode, you can use several functions, such as skew and datum, to set up a job. You can also use this as a normal DRO when you use your machine manually.

Move Table

The move table feature lets you move the table rapidly (or at an established feed rate) using the arrow keys.

- Press the **MOVE TABLE** softkey to turn the servo motors on. Press it again to turn them off.

- Enter the desired feed rate or skip this step to move at a rapid feed rate. Press the **CLEAR** key to enter a new feed rate.

- If you want the arrow keys to move the table in increments, press the **0.001**, **0.01** or **0.1** softkey. (A different set of softkeys will appear if you are measuring in millimeters.)

- Move the table. You can move the X- and Y-axes simultaneously by pressing two arrow keys at the same time.

- Press the **FEED+** and **FEED-** keys to adjust your feed rate.

It's a good idea to fold in the handles before moving the table.

Moves Table Away From You

Moves Table Left

Moves Table Right

Moves Table Towards You
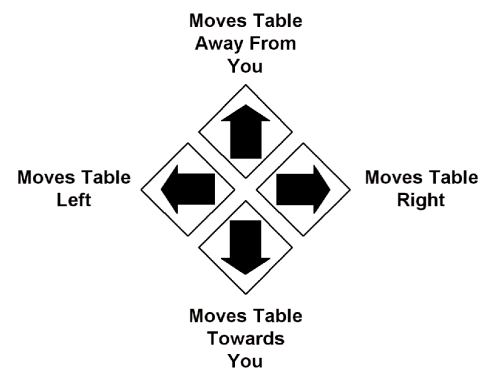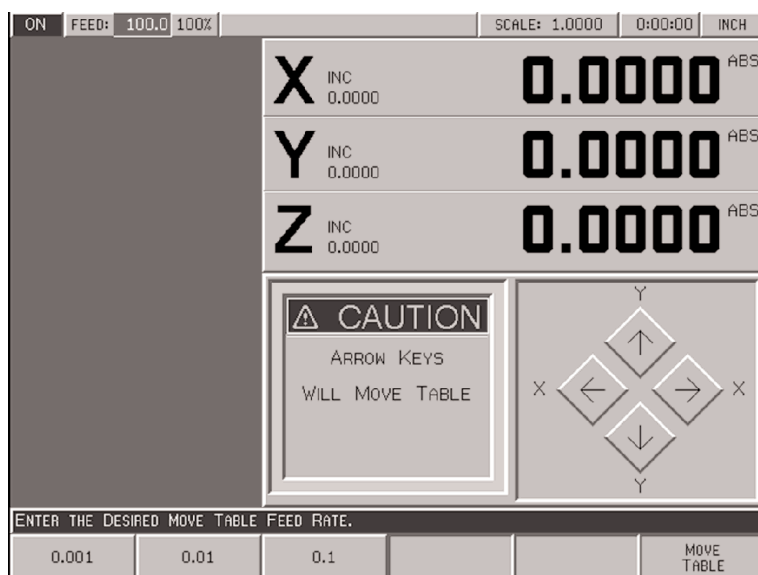
Teach Position

Whenever an X, Y or Z coordinate is being entered the **TEACH POSITION** softkey will be available. Pressing this key will place the current absolute position in the coordinate entry field.

To "teach" **MILL*PWR*** a coordinate (while programming a milling function, such as a line):

> *As you are programming or using a milling function, you can switch between the DRO and PGM screens without losing the program.*

- Using the arrow keys, highlight the X-, Y- or Z-axis field.

- Move the tool, indicator, or electronic edge finder to the position you want to teach.

- Press the **TEACH POSITION** softkey to enter that location and press **ENTER**. (With an electronic edge finder, the points will automatically be entered on contact—even if you over-travel.) See Using An Electronic Edge Finder.

- Repeat the steps above for each axis and each location you want to teach.

- Press **USE** to accept the information or press **CANCEL** to return to the previous screen without saving the information.

Using an Electronic Edge Finder

An ACU-RITE® Electronic Edge Finder can "teach" positions, find the center point of a circle, skew a part or locate datum (also known as workpiece zero) by simply "touching off" on the part. The greatest advantage of an electronic edge finder is that it instantly senses when contact is made allowing for over-travel.

MILL*PWR* SETUP PARAMETERS define the "Diameter" and "Unit" of measure (either inches or millimeters) for an electronic edge finder. Once this information has been entered, **MILL*PWR*** will automatically compensate for the radius of the tip of the electronic edge finder when performing any of the operations mentioned above.

<u>To define the diameter and unit of measure:</u>

• Press the **SETUP** key and highlight "Electronic Edge Finder."

• Press the **ENTER** key.

• Enter a value for the edge finder's diameter and unit of measure (inches or millimeters), then press the **USE** key.

• Press the **USE NEW SETTINGS** softkey.

To teach **MILL*PWR*** a coordinate, highlight the appropriate field then slowly move the table until the electronic edge finder touches the workpiece. When the electronic edge finder touches the part's surface, the coordinate will appear in the data field. The electronic edge finder is active whenever an entry field is highlighted.

Skewing a Part

Save time setting up a job by skewing the part. The skew function automatically compensates for the angle offset of thepart. If a part is not perfectly parallel with either the X- or Y-axis, indicating it in is not necessary.

To skew a part, simply "touch off" on two or more points along one axis (either X or Y). Use an electronic edge finder to skew a part, or use a mechanical indicator and teach position—either way, it's fast and easy.

Note: Choose a line to make parallel with the table's X- or Y-axis—do not enter coordinates along a curve, along two different lines, or along a line that's positioned at a true 45 degree angle. **MILL*PWR*** will calculate the skew angle based upon a straight line between the points entered.

If you are working with a part that has a rough edge, it's best to enter multiple points along the straightest edge so that **MILL*PWR*** can more accurately compensate for the skew angle.

Note: When two or more points are entered in the skew calculator **MILL*PWR*** will calculate the angle and adjust the X and Y plane. Angles that are less than 45° are adjusted down to the X-axis, and angles greater than 45° adjust up to the Y-axis.

Note: The skew feature does not not work with G-code programs. Remove any skew angle prior to running a G-code program.

To skew a part or vise:

Using an electronic edge finder

- Press the **SKEW** softkey.

- Touch off on two or more points along any single straight edge
  of the part. Notice the "Points" and "Angle" change as
  points are entered.

- Press the **USE** key to accept all of the points and return to the
  DRO screen. Press the **CANCEL** key to return to the DRO screen
  without accepting any points or affecting the previous skew
  angle.

  The **CLEAR ANGLE** softkey will reset the number of points and
  the skew angle to zero.

Here a work-piece was used as an example. You can also touch off on a vise or fixture.

Using teach position:

- Press the **SKEW** softkey.

- Move the table so that a mechanical indicator rests against any straight edge
  on the part. Press the **TEACH POSITION** softkey to enter the coordinate.
  Notice that the "Points" change.

- Now move the table so that the mechanical indicator touches another point on
  the same straight edge. Press the **TEACH POSITION** softkey. Notice that
  the "Points" and "Angle" change.

  Repeat this process for any additional points.

- Press the **USE** key to accept all of the points and return to the DRO screen.
  Press the **CANCEL** key to return to the previous screen *without* affecting the
  previous skew angle.

  The **CLEAR ANGLE** softkey will reset the number of points and the angle to zero.

Establishing Datum

Datum, also known as workpiece zero or absolute zero, is the point of reference that **MILL*PWR*** bases all of the part's coordinates from.

It is necessary to establish datum for every job. Datum's location may be indicated on a print; if it's not, establish a datum that enables most of the part's dimensions to be entered directly, with the least amount of calculations.

When establishing datum, you may find it easiest to locate a known point on each axis, such as the edge of the part or a location on the vise or fixture.

Refer to the procedure below as a basic guide for establishing datum. Decide to "touch off" using an electronic edge finder, a mechanical edge finder or a tool. Datum may be set at a point on the top surface or a position above or beneath the surface. X and Y datum may be set on an edge, or offset into or off of an edge, or where there's no material present (such as in the center of a circular part). There are many possibilities-- do what's easiest for a particular job.

MILL*PWR* will retain datum even after the system has been powered down. (See Finding Home).

To establish datum:

Where and how datum is establish will vary from job to job. One of the most common methods of establishing datum is described below. Apply the same principles when setting datum for future jobs, making adjustments to the procedure as needed.

The following example will establish datum at the corner where the left, front and top surfaces of the part intersect. This is accomplished by "touching" each face with the tool used to cut the part.

Define datum one axis at a time. Begin here with the X-axis:

- From the DRO screen, press the **DATUM** softkey.

- Insert the proper tool into the spindle.

- Position the tool so that it is near, but not touching, the left side of the part.

- Lower the tip of the tool so that it falls below the top surface of the part.

- Move the table along the X-axis slowly, while spinning the tool by hand. Pay close attention as the tool approaches the part— a subtle bump will be felt when they come into contact. **Stop the table at the moment the tool touches the part.**
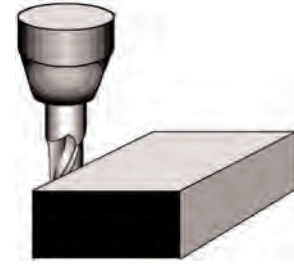
- Using the keypad, enter the radius of the tool (the distance from the center of the tool to the edge of the part). Be sure to specify if it's a negative value.

    Note: For this example, specify a negative value, because the tool's center is on the negative side of datum (refer to **Axis Conventions**).

- Press the **ENTER** key.

Use the MOVE TABLE softkey to help with long table moves.

Now set datum for the Y-axis using the same procedure:

- Position the tool so that it is near, but not touching, the front face of the part. The tip of the tool should fall below the top surface of the part.

- Move the table along the Y-axis, slowly spinning the tool by hand. Pay close attention as the tool approaches the part— A subtle bump will be felt when they make contact with each other. **Stop the table at the moment the tool touches the part.**

- Using the keypad, enter the radius of the tool into the"Y" field (be sure to specify if it is a negative value).
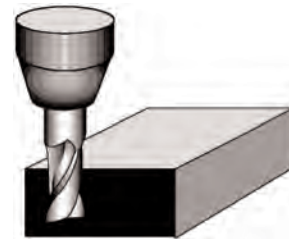
- Press the **ENTER** key.

Set datum for the Z-axis:

- Position the tool so that its tip touches the top surface of the part.

- Using the keypad, enter "0" into the "Z" field (or press the Z = 0 softkey).

- Press the **ENTER** key.

- Press the **USE** key.

Datum has now been established for X, Y, and Z

You can quickly move to Datum for X and Y by pressing the POS key. Check that your position is 0.0000 ABS for both X and Y, then press the GO key.

It's a good idea to test the datum setting before programming. To confirm that the new datum is correct:

- Raise the tool and move the table until both the X- and Y-axes displays read "0.0000."

- Lower the tool until it touches the part.

- Check the tool's position—the lower left corner of the part should be directly beneath the center point of your tool.

- Now check the readout. If the Z-axis display says "0.0000," then the datum is accurate. If a value other than 0.0000 appears in the Z-axis display, repeat the procedure for establishing datum.
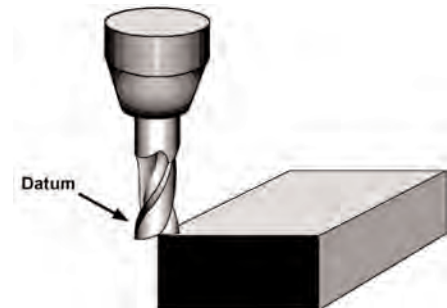
Hard Key Milling Functions

Most of the hard key milling functions can be used
individually as one time milling routines without creat-
ing a program.

The one hard key milling function not used as
a one-time milling function is **BLEND**. The blend func-
tion inserts a connecting radius between two features
(steps) in a program.
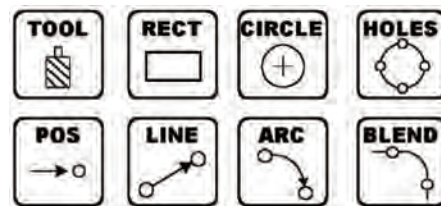
Hard key milling functions are ideal for jobs that only require one operation. Provide the
required information once, and **MILL*PWR*** will "remember" it for each piece machined.

Example:

> To drill the same bolthole pattern on several identical parts. Instead of creating a one-step
> program, save time by using the **HOLES** hard key milling function from the DRO screen.
>
> • First, set up a tool and workpiece.
>
> • From the DRO display, press the **HOLES** key, and select which hole pattern you
>   want to drill from the available softkeys—**ROW, FRAME, ARRAY,** or **BOLT CIRCLE**.
>
> • Enter the required information and press the **GO** key to machine the first part.
>
> • Change the part. Press the **HOLES** key, then the pattern.
>
> **MILL*PWR*** will automatically refer to the prior data entered for each part thereafter.

To change the hole pattern size, location or number of holes, press the **HOLES** key again. Now press the appropriate softkey, enter the new information and then press the **GO** key.

This may also be applied to rectangles, circles, lines and arcs. The rectangle and circle milling functions require you to establish a tool offset. Lines and arcs only require a tool offset if the tool follows a left or right offset. It's a good idea to set up a tool before using either of these function keys. Refer to **Program Steps** for a complete description of each function.

Note: The "Tool" setting on **MILL*PWR***'s status bar (located at the top of the display) will indicate which tool has been selected. If the "Tool" status bar is blank, no tool has been selected.

PROGRAMMING

Programming Considerations

"From" and "To" Points

Lines and arcs are defined by their "From" point
(the point where they begin) and "To" point (the
point where they end).
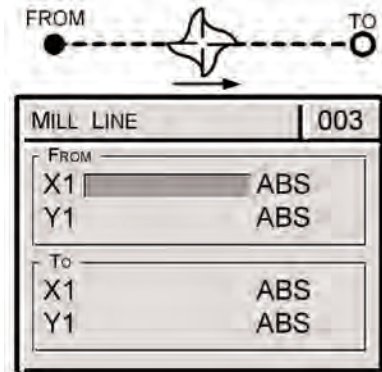
Depth of Cut

Since the Z-axis is not controlled by a servo motor, make changes to the depth
of cut manually.

If the Z-axis depth is programmed, **MILL*PWR*** will preset the programmed value into the
DRO. When it's time, **MILL*PWR*** will display the DRO screen and ask you to position the
depth.

If you are not programming a depth, leave it set to 0.

Tool Offset

By using left and right offsets, dimensions can be pro-
grammed as identified on the print.
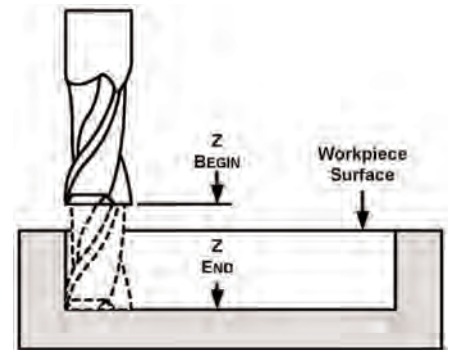
When a line, arc, frame, etc., is programmed, use the "Tool Offset" field to tell **MILL*PWR***
which side of the part the tool should be cutting on.

To determine which offset to use, picture yourself standing behind the tool as it is moving.
If the tool is on the left side of the workpiece, use a "left" offset. If the tool is on the right
side of the workpiece, use a "right" offset.

With a "center" offset, the programmed dimensions are for the center of the tool.

For some milling functions, like frame and arc, "inside" and "outside" offsets are available to make it easier to define a tool offset.

LEFT OFFSET

RIGHT OFFSET
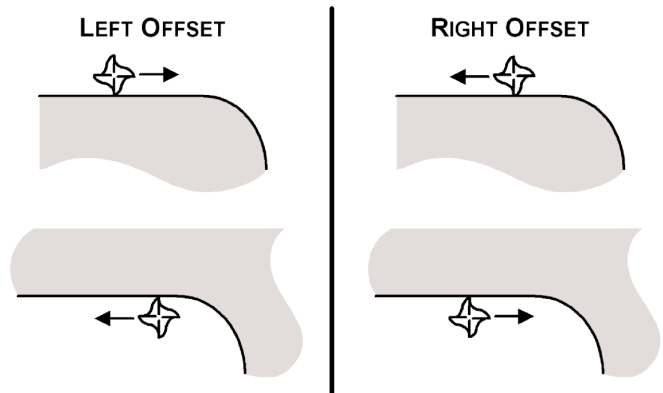
Datum Selection

Datum is where workpiece (absolute) zero is located. If datum isn't defined on the print, then determine datum based upon where most of the dimensions originate. Pick a point which will allow most of the dimensions to be entered directly, with few (if any) calculations (refer to **Establishing a Datum**).

Absolute vs. Incremental Dimensions

MILL*PWR* allows both absolute and incremental dimensions. A dimension measured from the point defined as datum is an absolute dimension. A dimension measured from any other point is an incremental dimension.

In the example below, the print on the left shows datum located at the center of Hole F—all dimensions are absolute.

The print on the right shows datum located in the lower left corner—point A. Most of these dimensions are incremental.
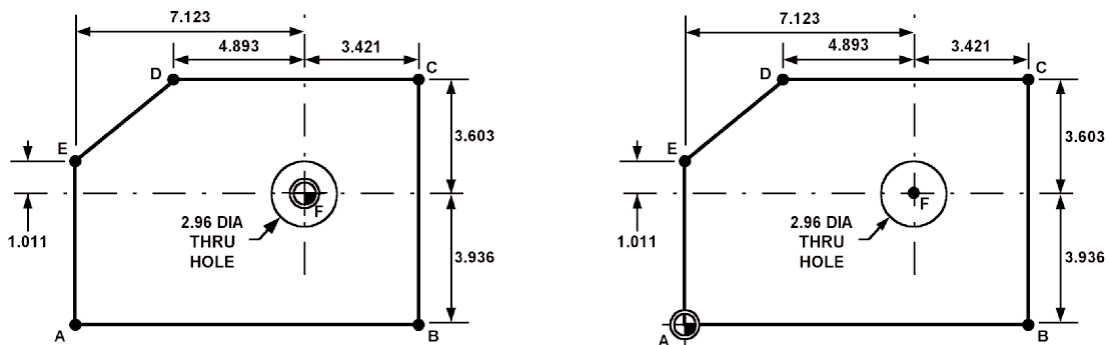
Continuous Milling

MILL*PWR* will cut a continuous contour of lines and/or arcs, without stopping. **MILL*PWR*** will automatically rec- ognize continuous contours as they are programmed. There are no special key presses or other functions to learn.
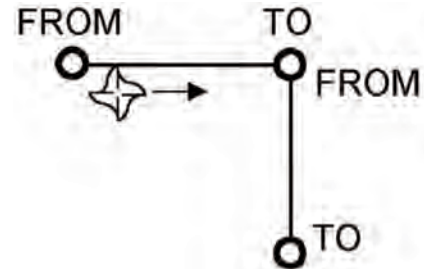
For lines and arcs to be continuous, they must:

- have the same depth
- be cut with the same tool
- be cut using the same tool offset
- share a common "From" or "To" point (one step must end at the point where next begins)

If one step follows another, **MILL*PWR*** assumes that they are to be connected. It auto- matically fills in the "From" point, "Depth," and "Tool Offset." To complete the step, fill in the "To" point and press **USE**.

Note: **MILL*PWR*** will allow different feed rates within each step of a contour.

Single lines indicate an open-ended continuous tool path.

```
        CURRENT PROGRAM
   001     SET TOOL
   002 ⌐ MILL LINE
   003 |  MILL ARC
   004 |  MILL LINE
   005 |  MILL LINE
   006 |  MILL LINE
   007 |  BLEND
   008 L MILL LINE
```

Double lines indicate a closed con- tinuous tool path.

```
        CURRENT PROGRAM
   001     SET TOOL
   002 ⊓ MILL LINE
   003 || MILL ARC
   004 || MILL LINE
   005 || MILL LINE
   006 || MILL LINE
   007 || BLEND
   008 || MILL LINE
   009 ⊔ MILL LINE
```

Note: An "X" after a step number indicates that the step is invalid. Highlight the step, press enter and correct the information as needed. Press use when finished.

Creating a Program

- Press the **PGM** key, and the following program screen will appear.

Programs are created by developing a list of milling steps to be performed. As steps are added, each step will immediately be drawn on the screen displaying a graphic representation of the part in progress.

- To enter a milling step, press the appropriate hard key or softkey milling function (such as Tool). The milling function keys are the eight yellow keys located in the upper right-hand corner of your keypad. When pressed, the function selected will appear in the program listing and the form opens allowing the information describing the step to be entered.

Program steps are added here.

Enter information about a milling function such as an arc into windows like this.
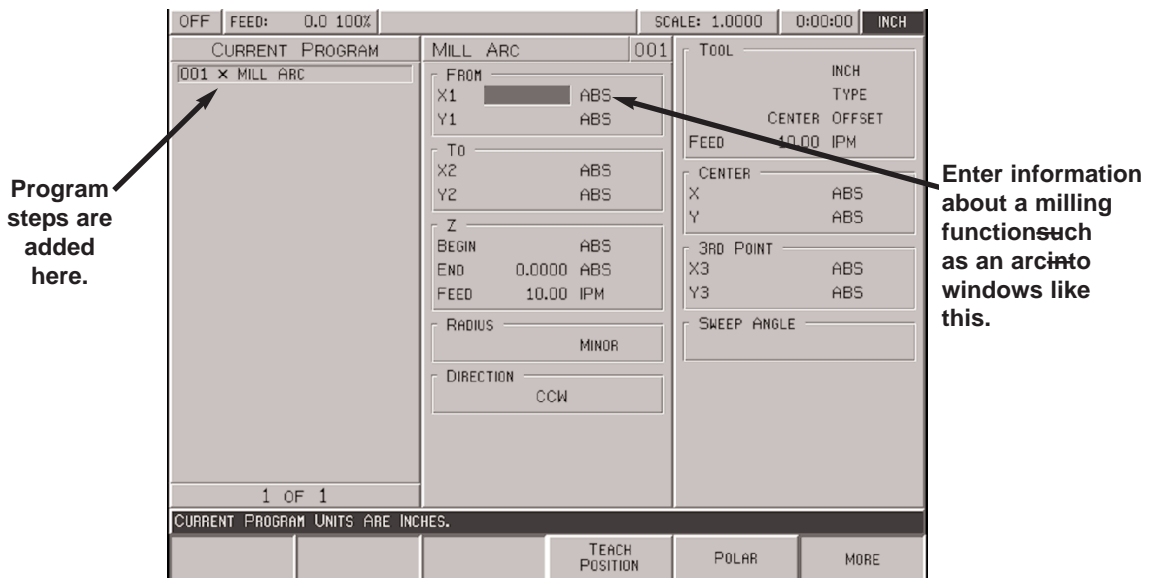
- After entering all the data for a step, press the **USE** key to add the step to the program. This immediately updates the part graphic and positions the cursor for the next step.

To exit a milling function without completing it, simply press the **CANCEL** key.
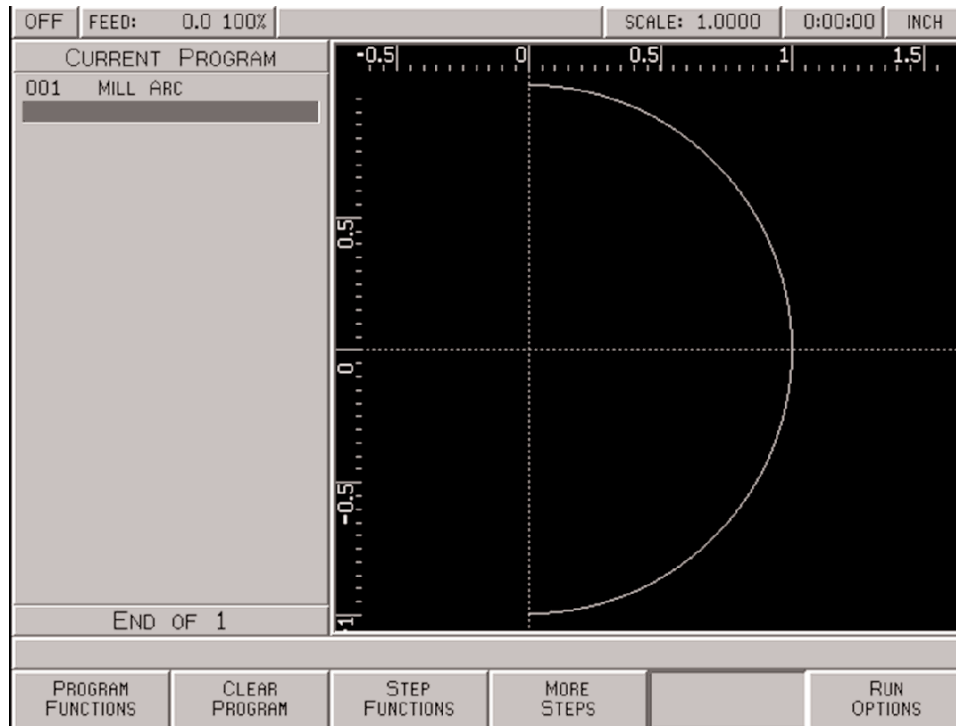
- To edit a step, use the arrow keys to highlight the step and press **ENTER**. After making changes, press **USE** to accept the changes.

- To delete a step, highlight the step to be deleted, then press **CLEAR**.

- To insert a step between the two existing steps, position the cursor on the lower step, and press the desired milling function key.

In addition to the milling functions, the **MORE STEPS** softkey displays a number of other useful steps, such as **CUSTOM POCKET**, **ISLAND, MIRROR**, **REPEAT** and **ROTATE**—each of which are described in the **Program Steps** section of this manual.

The View Key

To see part graphics in more detail, press the **VIEW** key to access the following softkeys:

FOLLOW TOOL	SHOW TOOL PATH		ZOOM IN	ZOOM OUT	RESTORE

FOLLOW TOOL

The **FOLLOW TOOL** softkey will automatically adjust the part graphic so that the tool is always in view during Dry Run and the execution of the program.

SHOW TOOL PATH

The **SHOW TOOL PATH** softkey shows the tool's cutting path during the Dry Run and execution of a program. With this feature enabled, the toolpath remains highlighted.

ZOOM IN, ZOOM OUT and RESTORE

The **ZOOM IN** softkey will magnify the part graphic. The arrow keys will adjust the view, up, down, left and right. The **ZOOM OUT** softkey will de-magnify the part graphic. **RESTORE** will return the part graphic to its original size.

Tip: Use **FOLLOW TOOL**, **SHOW TOOL PATH** and **ZOOM** functions simultaneously.

Press the **VIEW** key (or the **CANCEL** key) to return to the PGM screen.

Running a Program

There are a few things that may need to be done before running a program, such as skewing the part and establishing datum.

Skewing a Part

Note:　It is important to skew a part prior to establishing datum for accuracy.

Save time setting up a job by skewing the part. The skew function automatically compensates for the angle offset of the part. If a part is not perfectly parallel with either the X- or Y-axis, indicating it in is not necessary.

To skew a part, simply "touch off" on two or more points along one axis (either X or Y). Use an electronic edge finder to skew the part, or use a mechanical indicator and teach position—either way, it's fast and easy.

Note:　Choose a line to make parallel with the table's X- or Y-axis—do not enter coordinates along a curve, along two different lines, or along a line that is positioned at a 45 degree angle. **MILL*PWR*** will calculate the skew angle based upon a straight line between the points entered.

If you are working with a part that has a rough edge, it's best to enter multiple points along the straightest edge so that **MILL*PWR*** can more accurately compensate for the skew angle.
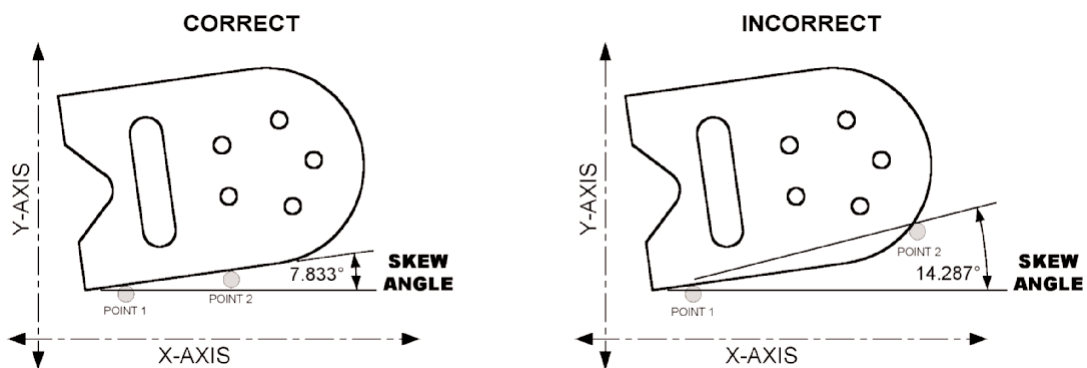
Note:　When two or more points are entered in the skew calculator **MILL*PWR*** will calculate the angle and adjust the X and Y plane. Angles that are less than 45° are adjusted down to the X-axis, and angles greater than 45° adjust up to the Y-axis.

Note:　The skew feature does not not work with G-code programs. Remove any skew angle prior to running a G-code program.

Using an electronic edge finder:

- From the DRO screen, press the **SKEW** softkey.

- Touch off on two or more points along any single underline straight edge of the part. Notice that the "Points" and "Angle" change as you enter points are entered.

- Press **USE** to accept all of the points and return to the DRO screen. Press **CANCEL** to return to the DRO screen *without* accepting any points or affecting the previous skew angle.

    The **CLEAR ANGLE** softkey will reset the number of points and the skew angle to zero.

Using teach position:

- From the DRO screen, press the **SKEW** softkey.

- Move the table so that a mechanical indicator rests against any underline straight edge on the part. Press the **TEACH POSITION** softkey to enter the coordinate. Notice that the "Points" change.

- Now move the table so that the mechanical indicator touches another point on the same straight edge. Press the **TEACH POSITION** softkey. Notice that the "Points" and "Angle" change.

    Repeat this process for any additional points.

- Press **USE** to accept all of the points and return to the DRO screen. Press **CANCEL** to return to the previous screen *without* affecting the previous skew angle.

    The **CLEAR ANGLE** softkey will reset the number of points and the angle to zero.

Establishing Datum

Datum, also known as workpiece zero or absolute zero, is a point of reference that **MILL*PWR*** bases all of the part's coordinates from.

It is necessary to establish datum will need to be establish for every job. Datum's location may be indicated on a print; if it's not, establish a datum enables most of the part's dimensions to be entered directly, with the least amount of calculations.

When establishing datum, locate a known point on each axis, such as the edge of the part or a location on a vise or fixture.

Refer to the example below as a basic guide for establishing datum. Decide to "touch off" using an electronic edge finder, a mechanical edge finder, or a tool. Datum may be set at a point on the top surface or a position above or beneath the surface. X and Y datum may be set on an edge, or offset into or off of an edge, or where there's no material present (such as in the center of a circular part). There are many possibilities, do what is easiest for the particular job.

MILL*PWR* will retain the datum you've set after your system has been powered down. (See **Finding Home**).

To establish datum:

Where and how datum is established will vary from job to job. One of the most common methods of establishing datum is described below. Apply the same principles when setting datum for future jobs, making adjustments to the procedure as needed.

The example below will establish datum at the corner where the left, front and top surfaces of the part intersect. This is accomplished by "touching" each face with the tool used to cut the part.

Define datum one axis at a time. Begin with the X-axis:

• From the DRO screen, press the **DATUM** softkey.

• Insert the proper tool into the spindle.

- Position the tool so that it is near, but not touching, the left side of the part.

- Lower the tip of the tool so that it falls below the top surface of the part.

- Move the table along the X-axis, while slowly spinning the tool by hand. Pay close attention as the tool approaches the part—a subtle bump will be felt when they come into contact. **Stop the table at the moment the tool touches the part.**

Use the MOVE TABLE softkey to help with long table moves.

- Using the keypad, enter the radius of the tool (the distance from the center of the tool to the edge of the part) into the "X:" field. Be sure to specify if it's a negative value.
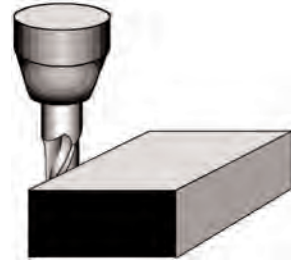
  Note: For this example, specify a negative value, because the tool's center is on the negative side of datum (refer to **Axis Conventions**).

- Press the **ENTER** key.

Now set datum for the Y-axis using the same procedure:

- Position the tool so that it is near, but not touching, the front face of the part. The tip of the tool should fall below the top surface of the part.

- Move the table along the Y-axis, slowly spinning the tool by hand. Pay close attention as the tool approaches the part—a subtle bump will be felt when they come into contact. **Stop the table at the moment the tool touches the part.**

- Using the keypad, enter the tool's radius into the"Y:" field (be sure to specify if it is a negative value).
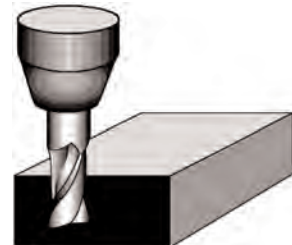
- Press the **ENTER** key.

Set datum for the Z-axis:

- Position the tool so that its tip touches the top surface of the part.

- Using the keypad, enter "0" into the "Z:" field (or press the Z = 0 softkey).

- Press the **ENTER** key.

- Press the **USE** key.

Datum has now been established for X, Y and Z.
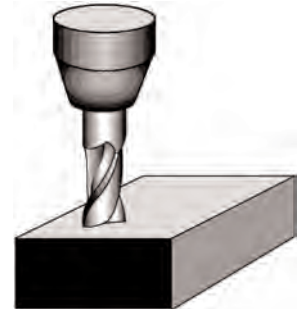
You can quickly move to datum for X and Y by pressing the POS key. Check that the go to position is 0.0000 for both X and Y, then press the GO key.

It's a good idea to test the datum setting before programming. To confirm that the new datum is correct:

- Raise the tool and move the table until both the X- and Y-axes displays read "0.0000."

- Lower the tool until it touches the part.

- Check the tool's position—the lower left corner of the part should be positioned directly beneath the center point of the tool.

- Now check the readout screen. If the Z-axis display says "0.0000," then the datum is accurate. If a value other than 0.0000 appears in the Z-axis display, repeat the procedure for establishing the datum.
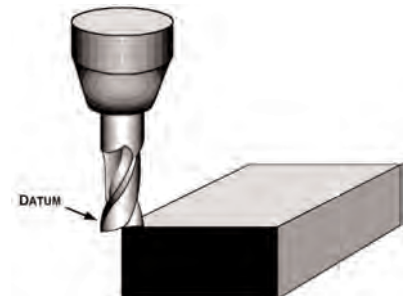
Testing Your MILL*PWR* Program

Before machining a part, it is always a good idea to test the program for things like the correct tool path, count direction, feed rate, and sequence of operations. **MILL***PWR* provides several run-time options to provide assistance. From the PGM screen, press **RUN OPTIONS** to display the following softkeys:

SINGLE STEP	DRY RUN	GRAPHICS ONLY	MANUAL POSITIONING	DISABLE LOOK AHEAD	RUN OPTIONS

Press any softkey to activate the option; press it again to deactivate it.

SINGLE STEP

Normally, a continuous contour will be machined without stopping. With "Single Step" activated, **MILL***PWR* will stop after each step. Use this feature to check the position of the tool relative to the part and ensure that the tool path and other program details are correct.

DRY RUN

With "Dry Run" activated, **MILL***PWR* will run the program at high speed without stopping. Follow the sequence of steps and visually follow the position of the tool relative to the part to ensure that the tool path and other program details are correct.

To test the program quickly, press the DRY RUN and GRAPHICS ONLY softkeys.

The dry run speed is defined in "Setup." A prompt asking to raise the tool is displayed before the dry run begins.

GRAPHICS ONLY

With this activated, the table does not move, but the graphics will show how the part will be cut. Use this feature to see all the normal feed rates, tool changes and so on.

Tip: Dry Run and Graphics Only can be used to quickly verify your program.

MANUAL POSITIONING

Use this option to position the table using the handles. **MILL*PWR*** will operate just like a pro-grammable readout—each target position will be preset into the readout, and you will be prompted to position the table manually. This feature is especially useful when navigating around islands.

DISABLE LOOK AHEAD

Normally, **MILL*PWR*** checks each step in a continuous tool path with other steps to determine if there is an intersection in the tool's cutting path (such as a figure 8). This is called look ahead. Press the **DISABLE LOOK AHEAD** softkey, the program will run *without* performing this function.

Pressing the GO Key

> ⚠ Whenever you are about to run a program, check that the handles are recessed.

To run a program, with or without any of the run options, highlight the step to begin with and press **GO**. **MILL*PWR*** will automatically pause at points that require action (e.g., change tools, raise the quill, etc.) After the task has been completed, press the **GO** key again, and the table will move to the next position. At this point, you will be prompted to drill or set the tool to the programmed depth.

Before any rapid move, **MILL*PWR*** will display a warning message indicating that the table is about to move at high speed. At this time, check that the tool is clear of the workpiece and fix-turing. Pressing the **GO** key, confirms that the tool is clear and ready for the table to move.

The remote STOP/GO switch acts as PAUSE if the table is moving and as GO if the machine is paused or stopped.

ATTENTION

RAPID MOVE

RAISE QUILL

PRESS 'GO'

Pressing the **STOP** key once, causes the tool to pause in its cutting path, and the following prompt will appear:

To move quickly to a step, key in its number (look in the message bar), and press ENTER.

ATTENTION

PAUSE

PRESS 'GO'

Press **GO** to resume machining, or **STOP** again to end the program execution. To restart a program, move to the step you wish to begin with and then press **GO**.

Note: If starting in the middle of a program and the tool programmed is not displayed in the current tool field, start the program at the programmed tool step for that tool.

Feed+ and Feed-

The **FEED+** and **FEED-** keys will change the feed rate by a certain percentage with each key press. The feed rate percentage will be displayed in the status bar at the top of the screen. A feed rate percentage of 100% means that actual feed rates will run at 100% of the programmed feed rates. If the feed rate percentage is 50%, actual feed rates will run at half of the programmed feed rates.

Press the **FEED+** and **FEED-** keys at any time, even while the table is moving.

Machining to Zero

MILL*PWR* is factory preset in a distance to go display view. Any dimension programmed will be "preset" into the readout display. Every move will begin at the preset value and end at zero.

If a depth has been programmed, a prompt will instruct you to position the Z-axis. **MILL*PWR*** will preset the value into the readout's Z-axis. The DRO screen will appear, along with the following prompt:

```
ATTENTION
Z AXIS


SET TO
0.0000


PRESS 'GO'
```

- Position the Z-axis until the absolute display is 0.0000.

To set up **MILL*PWR*** to the incremental travel display view refer to **System Setup**. In this view, every move will begin at zero and end at the programmed position. For example, a programmed depth of -0.5000, will display, the following prompt:

```
ATTENTION
Z AXIS

SET TO
-0.5000


PRESS 'GO'
```

- Move the Z-axis until the absolute display shows the target position (e.g., -0.5000").

Program Functions

Accessing Load, Save, Delete, Merge, Backup and Directory Options

MILL*PWR* offers several versatile features for loading, saving, deleting, merging and backing up programs. Use these features to create directories to easily organize your programs.

To access these features, from the PGM screen, press the **PROGRAM FUNCTIONS** softkey.
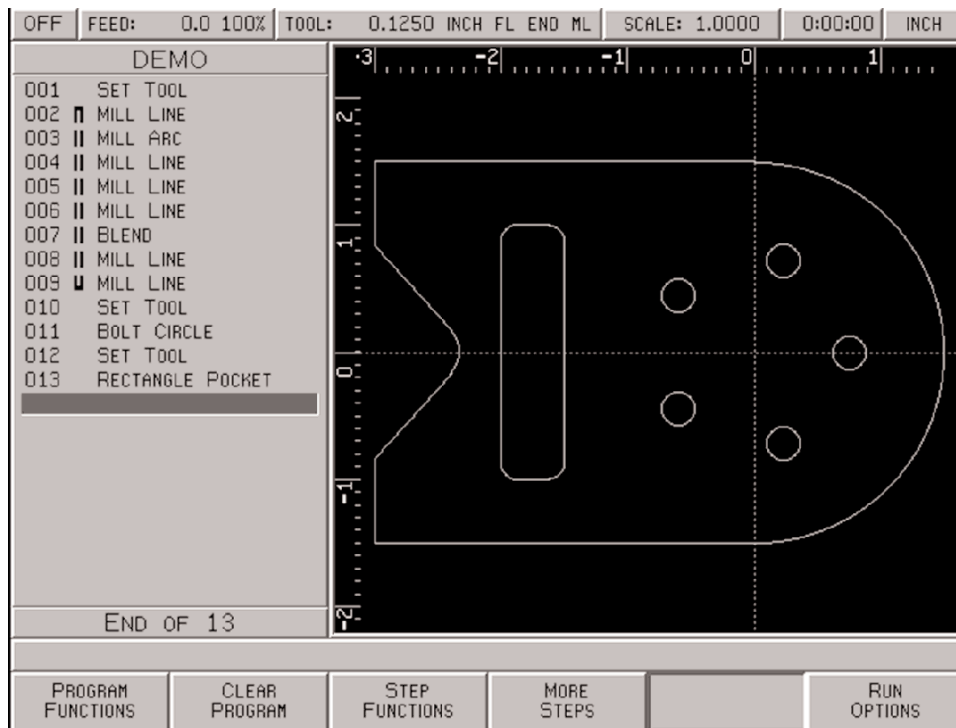
Use these keys to load a program, save the program, delete programs, merge a saved program into an open one, create a backup copy, or access, create or delete a directory.

Directories

One of the best ways to keep programs organized is to save them in directories. Directories are like file folders—they should be clearly labeled and contain closely related projects. They may be used to group programs by job, operator, date, customer, or any other method.

- Press the **PROGRAM FUNCTIONS** softkey, then press the **DIRECTORY** softkey.

SELECT A DIRECTORY FUNCTION.					
SELECT DIRECTORY	CREATE DIRECTORY	DELETE DIRECTORY			
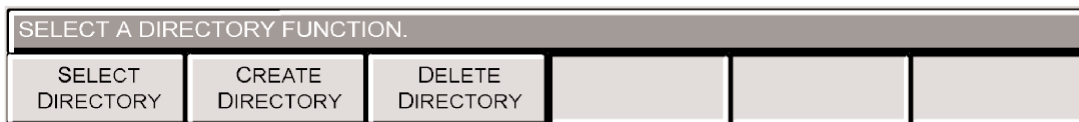
These keys can create a directory, open an existing directory, or delete a directory that you no longer need.

Note: A directory can only be deleted if it is empty. First delete or move all part programs from a directory prior to deleting it. See **Deleting A Directory**.

Creating a Subdirectory

The best approach to take when creating a subdirectory is to decide first where to place it. A directory can be placed on the"MILLPWR"directory, on a floppy disk ("A:") directory, or on a PC ("REMTSTOR") directory, or placed within other subdirectories.

In the example below, four subdirectories have been created to keep programs better organized. In this case, specific directories were created for three customers. "COMPANY1" has placed several part orders for a single month. To help find those part programs quickly and easily, they were saved in the subdirectory labeled "APRIL99."

We recommend creating matching directories on MILLPWR and your PC. This will make it easier to keep your original programs and backup programs organized.
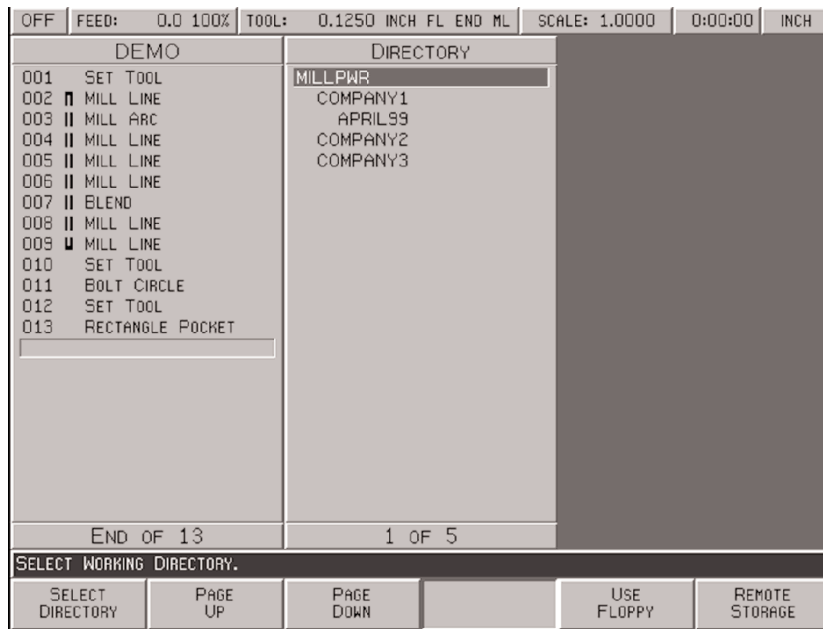
The subdirectories COMPANY1, COMPANY2 and COMPANY3, were created with MILLPWR as the main directory.

When the APRIL99 subdirectory was created, COMPANY1 was selected as the directory. Another subdirectory can be created under APRIL99, another one under that, one under that, and so on.

To create directories for your programs**:**

- Press the **PGM** key, the **PROGRAM FUNCTIONS** softkey and then the **DIRECTORY** softkey. Now press the **SELECT DIRECTORY** softkey, and a directory list will appear.

- Highlight the directory where you want to store the new subdirectory.

    Select **MILL*PWR****'s *internal memory:*

    "MILLPWR" and any subdirectories should appear under the "Directory" heading.

    Select a 3¹/₂" floppy disk:

    Press the **USE FLOPPY** softkey. "A:" and any subdirectories should appear under the "Directory" heading.

    Select your PC:

    Press the **REMOTE STORAGE** softkey. "REMTSTOR" and any subdirectories should appear under the "Directory" heading.

- Using the arrow keys, highlight the existing directory you want to put the new subdirectory in. (In this example, we highlighted "MILLPWR," then created a subdirectory entitled "COMPANY1.")

- Press the **SELECT DIRECTORY** softkey again to verify your choice. The "Directory" screen will disappear.

- Now press the **DIRECTORY** softkey.

- Press the **CREATE DIRECTORY** softkey. You will be asked to name your directory.

- Name your directory using the numeric keys on the operator console or by selecting letters from the **ALPHABET** option. Use the arrow keys to move from letter to letter then press the **ENTER** key to make a selection. Program names are limited to eight characters, consisting of numbers and/or letters.

*The REMOTE STORAGE softkey will only appear if MILL**PWR** has been set up for remote storage.*

- After you have named your directory, press the **CREATE DIRECTORY** softkey again to enter your choice. The "Directory" screen will now disappear.

Additional directories (and subdirectories) may be added at any time.

IMPORTANT
Creating a directory does not mean that the directory is selected. If you plan to save the current program in the directory you just created, you must select the new directory first. Otherwise, the program will be saved in the last directory that was selected.

Selecting a Directory

The **SELECT DIRECTORY** softkey allows you to open any of the directories that you have previously created on **MILL*PWR***'s internal memory, on a 3^1/$_2$" floppy disk, or on your PC. You may use this feature any time you save or load a program.

- Press the **PROGRAM FUNCTIONS** softkey, then the **DIRECTORY** softkey. Press the **SELECT DIRECTORY** softkey—a directory list will appear.

- Indicate where the directory you want to select is located.

  On **MILL*PWR***'s *internal memory:*
  "MILLPWR" and any subdirectories that you have created should appear under the "DIRECTORY" heading.

  On a 3$^1/_2$" floppy disk:
  Insert the 3$^1/_2$" floppy disk containing the directory into the floppy disk drive (located in the lower right-hand corner on the front of the operator console) and press the **USE FLOPPY** softkey. "A:" and any subdirectories you have created should appear under the "DIRECTORY" heading.

  On your PC:
  Press the **REMOTE STORAGE** softkey. "REMTSTOR" and any subdirectories you have created will appear under the "DIRECTORY" heading.

- Using the arrow keys, highlight the directory you want to open. (If the list is long, use the **PAGE UP** and **PAGE DOWN** softkeys to scroll through the list more quickly.)

- Press the **SELECT DIRECTORY** softkey again. The "DIRECTORY" list will disappear, indicating that your directory has been selected.

You can now save your program in the directory you have chosen (refer to **Saving a Program**); or if you prefer, load an established program from the directory you selected (refer to **Loading a Program**).

Deleting a Directory

MILL*PWR* will not delete directories that contain programs. First delete each program and subdirectories stored within the directory (refer to **Deleting a Program**).

To delete a directory:

- From the PGM screen, press the **PROGRAM FUNCTIONS** and **DIRECTORY** softkeys, then press the **DELETE DIRECTORY** softkey.

- Identify where the directory you want to delete is located.

    *On **MILL*PWR*'s internal memory:*
      "MILLPWR" and any subdirectories you have created should appear under the "DIRECTORY" heading.

    On a 3$^1/_2$" floppy disk:
      Insert the 3$^1/_2$" floppy disk containing the directory into the disk drive (located in the lower right-hand corner on the front of the operator console) and press the **USE FLOPPY** softkey. "A:" and any subdirectories you have created will appear under the "DIRECTORY" heading.

    On your PC:
      Press the **REMOTE STORAGE** softkey. "REMTSTOR" and any subdirectories you have created will appear under the "DIRECTORY" heading.

- Using the arrow keys, highlight the directory you want to delete.

- Press the **ENTER** key. Press the **YES** softkey to erase the directory or the **NO** softkey to cancel.

Saving a Program

You can save programs in any of three places—on **MILL*PWR***'s internal memory, on a 3^1/$_2$" floppy disk, or on a PC. It is always a good idea to save your programs often to avoid losing valuable information.

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then select the directory where you want to save your program (refer to **Selecting a Directory** and/or **Creating a Directory**).

- Return to the PGM screen, then press the **PROGRAM FUNCTIONS** and **SAVE** softkeys.

```
OFF | FEED:     0.0 100% | TOOL:   0.1250 INCH FL END ML | SCALE: 1.0000 | 0:00:00 | INCH

          DEMO                    SAVE PROGRAM
001    SET TOOL           ┌ PROGRAM NAME ─────────┐
002 ⊓ MILL LINE           │   DEMO                │
003 ‖ MILL ARC            └───────────────────────┘
004 ‖ MILL LINE
005 ‖ MILL LINE
006 ‖ MILL LINE
007 ‖ BLEND
008 ‖ MILL LINE
009 ⊔ MILL LINE
010    SET TOOL
011    BOLT CIRCLE
012    SET TOOL               ALPHABET
013    RECTANGLE POCKET
                          ┌ CHARACTERS ───────────┐
                          │                       │
                          │      A  B  C          │
                          │   D  E  F  G  H       │
                          │   I  J  K  L  M       │
                          │   N  O  P  Q  R       │
                          │   S  T  U  V  W       │
                          │      X  Y  Z          │
                          │                       │
          END OF 13       └───────────────────────┘
ENTER A NAME FOR THE PROGRAM.
  SAVE  |        | ALPHABET | FORMAT... | USE    | REMOTE
                              (MPT)     | FLOPPY | STORAGE
```

You will be asked to name your program (refer to **Naming a Program**).

- Press the **SAVE** softkey. The program's name should now appear in the left column above the program steps.

If you make any changes, make sure that you save your program again.

Naming a Program

Before saving a program, **MILL*PWR*** requires you to name it.

- If you want to use letters, press the **ALPHA-BET** softkey. An alphabet menu will appear just below the "Program Name" field.

- Using the arrow keys, you can move from one letter to the next. Press the **ENTER** key to select a letter.

To add numbers to your program name, simply press any of the number keys on the keypad.

You may choose up to eight characters for your program name, mixing numbers and letters if you wish.

- Press the **SAVE** softkey. **MILL*PWR*** will store your program in the directory you have selected.

A message will alert you if the program was not saved properly, or if the name that you have chosen already exists.

If you have accidentally selected the wrong letter or number, press the CLEAR key and rename your program.

```
SAVE PROGRAM

PROGRAM NAME
┌─────────────┐
└─────────────┘


ALPHABET
CHARACTERS
    A B C
   D E F G H
   I J K L M
   N O P Q R
   S T U V W
     X Y Z
```
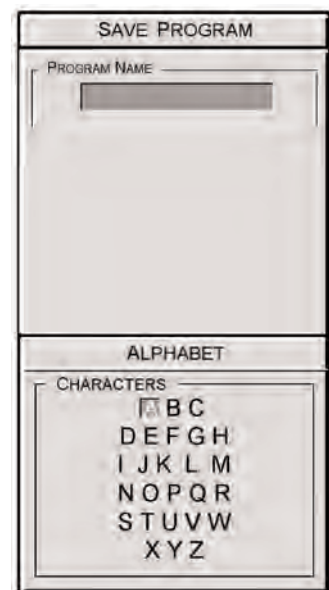
Loading a MILL*PWR* (MPT) Program

The **LOAD** softkey allows you to open programs that have already been saved. The steps below tell you how to load a program from **MILL*PWR***'s internal memory, a $3^1/_2$" floppy disk, or your PC.

From **MILL*PWR***'s internal memory:

- Save, then clear any open programs.

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then press the **LOAD** softkey. The last directory that was selected and any programs it contains will appear.

*If the program is saved in the **MILL*PWR*** directory:*

- "MILLPWR" should appear at the top. If it doesn't, check that the **USE FLOPPY** and **REMOTE STORAGE** softkeys are not selected.

- Press the format button until MPT is displayed.

- Using the arrow keys, highlight the program you want to load, then press the **LOAD** softkey. The program you have selected should appear on the screen.
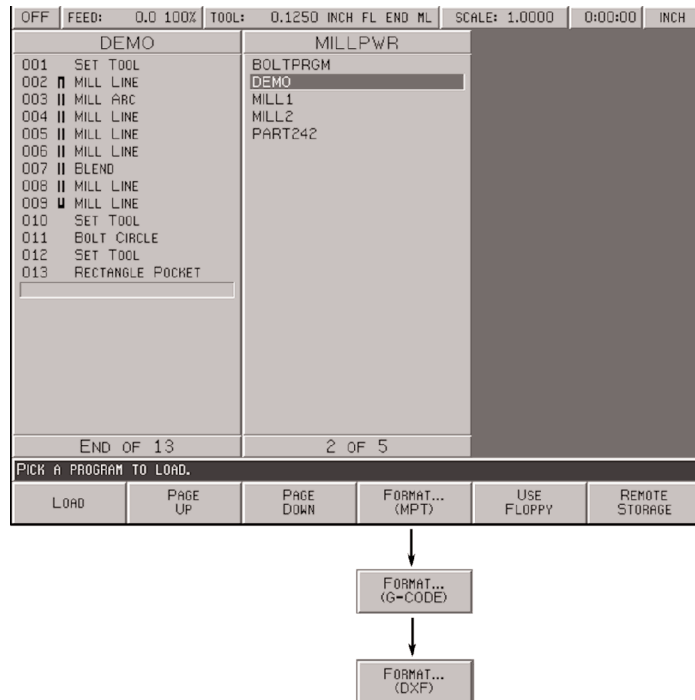
If the program is saved in a different directory:

• Press the **CANCEL** softkey.

• Press the **PROGRAM FUNCTIONS** softkey.

• Press the **DIRECTORY** softkey.

• Now press the **SELECT DIRECTORY** softkey.

• Using the arrow keys, highlight the directory that contains the program you want to load.

• Press the **SELECT DIRECTORY** softkey again. The "DIRECTORY" screen will disappear. Now that you have selected the appropriate directory, you can load the program.

• Press the **LOAD** softkey. The directory you chose should appear.

• Highlight the program you want to load, then press the **LOAD** softkey. The program you selected should now appear on the screen.

From a 3^1/2" floppy disk:

• Save, then clear, any open programs.

• From the PGM screen, press the **PROGRAM FUNCTIONS** softkey.

• Insert the 3^1/2" floppy disk containing the file you want to load into the floppy disk drive. Located in the lower right-hand corner of the operator console.

• Press the **DIRECTORY** softkey and then press the **SELECT DIRECTORY** softkey.

• Press the **USE FLOPPY** softkey. The "A:" directory and any sub-directories should appear.

• Select the directory that contains your program and then press the **USE** softkey.

• Press the **LOAD** softkey.

• Press the format key until MPT is displayed.

• Using the arrow keys, highlight the program you want to load. Press the **LOAD** softkey. The program you selected should now appear on the screen.

From your PC:

- Save, then clear, any open programs.

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then press the **DIRECTORY** softkey.

- Press the **SELECT DIRETORY** softkey.

- Press the **REMOTE STORAGE** softkey. (If the **REMOTE STORAGE** softkey does not appear, then it is likely the **MILL*PWR*** and your PC have not been set up. Refer to **Remote Storage** and/or **Installation Setup**.) Select the directory you wish to use.

- Press the **USE** softkey.

- Press the **LOAD** softkey.

- Press the **FORMAT** key until MPT is displayed.

- Select the directory that contains your program (refer to **Selecting a Directory**).

Translating a DXF file:

- Save, then clear, any running programs.

- Locate the directory containing your DXF file (refer to **Selecting a Directory**).

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then press the **LOAD** softkey.

- Select the storage location of your DXF file. (on a $3^{1}/_{2}$" floppy disk or on a PC):

If your DXF file is stored on a $3^{1}/_{2}$" floppy disk:

- Insert the floppy disk containing the DXF file you want to load into the floppy disk drive (located in the lower right-hand corner of the operator console).

- Press the **USE FLOPPY** softkey.

- Press the **FORMAT** softkey until DXF is displayed. The screen will display all of the DXF files stored in the directory you have chosen.

- Using the arrow keys, highlight the DXF file you wish to translate.

- Press the **LOAD** softkey.

If your DXF file is stored on a PC:

- Press the **REMOTE STORAGE** softkey. A "REMTSTOR" directory and any programs it contains should appear.

If the **REMOTE STORAGE** softkey does not appear, then it is likely that **MILL*PWR*** and your PC have not been set up properly (refer to **Remote Storage** and/or **System Setup**).

- Press the **FORMAT KEY** softkey until DXF is displayed. The screen will display all of the DXF files stored in the directory you've chosen.

- Using the arrow keys, highlight the DXF you wish to translate.

- Press the **LOAD** softkey.

MILL*PWR* will read the DXF files you selected, then translate the file's lines, points, arcs, and circles into the appropriate **MILL*PWR*** milling function steps. Default values will be assigned for any information that's missing from the DXF file (such as tool offset, feed rate, etc.).

MILL*PWR* will then arrange the steps in a logical order (based upon common end points) and create a tool path. After the tool path has been determined, the program will appear on the screen.

Test your program before machining to ensure that the program steps and tool path do what you want them to do. You can easily edit and rearrange steps as needed (refer to **Step Functions**).

Loading a G-Code file:

- Save, then clear, any running programs.

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey.

- Select the directory containing your program (refer to **Selecting a Directory**).

- Press the load softkey.

- Press the format softkey until G-Code is displayed.

- Using the arrows, highlight the program you wish to load.

- Press the **LOAD** softkey.

If your G-code file is stored on a 3 ¹/₂" floppy disk:

- Insert the floppy disk containing the G-code file you want to load into the floppy disk drive (located in the lower right-hand corner of the operator console).

- Press the **USE FLOPPY** softkey.

- Using the arrow keys, highlight the G-code file you wish to load.

- Press the **LOAD** softkey.

If your G-code is stored on a PC:

- Press the **REMOTE STORAGE** softkey. A "REMTSTOR" directory and any programs it contains should appear.

If the **REMOTE STORAGE** softkey does not appear, then it is likely that **MILL***PWR* and your PC have not been setup properly (refer to **Remote Storage** and/or **System Setup**).

- Select the directory containing your program (refer to **Selecting a Directory**).

- Press the format key until G-code is displayed softkey. The screen will display all of the G-codes files stored in the directory you have chosen.

- Using the arrow keys, highlight the DXF files you wish to translate.

- Press the **LOAD** softkey.

Note: G-code files are "<u>run only</u>." No editing can be done to the file at the **MILL***PWR*. All editing should be done within your CAD/CAM program (refer to **MILL***PWR* **G-code Conventions**).

Running a G-Code Program

Considerations when creating a G-code Program

MILL*PWR* has the ability to read and execute Numerical Code (G-Code) files, how ever those files can not be edited from the controller. It is important to create and proof the G-code file before attempting to machine a part.

The use of CAD/CAM Software is strongly recommended.

Tool Offsetting
No programmed cutter compensation is used so the tool path should be based on the center and tip of the tool.   Do not establish tool length offsets within CAD/CAM software. For repeatable tooling, tool length offsets can be established in the tool library (Diameter offsets are not used).

Using the Tool Library
Each "T" block refers to the corresponding number in the Tool Library. For example, T1 will cause **MILL*PWR*** to retrieve the tool length offset from tool 1 of the Tool Library. **MILL*PWR*** will then offset the spindle by this amount. T2 will cause **MILL*P-WR*** to retrieve the tool length offset from tool 2 of the Tool Library, etc. To set up a Tool Library refer to Setup.

> **It is very important not to have any tool length offsets in the Tool Library if the tooling is not repeatable.**

Failure to maintain the Tool Library can cause unpredictable results. Verifying tool length offsets prior to program execution is strongly recommended.

Loading a Program

A G-code program can be loaded into **MILL*PWR*** in the same manner as **MILL*PWR*** programs. (Refer to **Loading a Program** for instructions on loading a G-code program.) With G-code files, there is no accompanying graphics. Only the program list is displayed. Once loaded, **MILL*PWR*** will indicate lines of code with invalid and unsupported blocks with an "X". If the line containing the code is highlighted, an error message also appears in the message line indicating that the line contains invalid code. The invalid code can be removed by disabling the corresponding functions in your CAD/CAM software and re-posting the G-code program. Simply removing the invalid code from the G-code program can cause unpredictable results and is not recommended.

Running a G-code Program

The tool path should be proofed prior to attempting to machine a part on **MILL*PWR***.

Most CAD/CAM Software packages have this ability. When a program has been proofed and loaded into **MILL*PWR***, lower the knee and dry run the program to verify the tool path and speeds and feeds are correct.

Starting and Stopping a G-code Program
Always start the program from a place in the program where the feed rate, X-, Y-, and Z-axis position are known, such as a tool step. Alternate starting points can be programmed by placing the proper code in the desired locations.

Pressing the **GO** button will cause **MILL*PWR*** to begin executing the current G-code program. Always insure the program step highlighted is an appropriate starting point.

When a program is running, pressing the **STOP** button or the remote pendant will cause the program and all axis motion to pause. Pressing the remote pendant switch again or the **GO** button will cause the program to resume. Pressing the **STOP** button a second time will halt the program execution.

MILL*PWR* G-Code Conventions

Table of G & M Codes

This table lists both the supported and unsupported codes. Non-supported codes are shown in gray.

Code	Description	Comment
D	Tool Diameter	**MILL*PWR*** does not support automatic cutter compensation. Specifying a tool diameter index while cutter compensation is in effect will generate a run-time error. Specifying a diameter index while cutter compensation is off has no effect and is ignored.
F	Set Feed Rate	The feed rate uses the current program units in effect (ipm or mmpm). The setting applies to current and subsequent blocks. The default is determined from **MILL*PWR***'s configuration setup.
G0	Linear Interpolation (Rapid)	These commands generate table/quill motion. The motion command applies to current and subsequent blocks containing at least one X, Y, or Z coordinate. The default motion command is a linear move at feed (G1).
G1	Linear Interpolation (Feed)	
G2	Circular Interpolation (CW)	
G3	Circular Interpolation (CCW)	
G4	Dwell	This command causes the system to pause for a specified period of time *prior* to any motion commands within the blocks. The period of time is determined by the P value.
G10	Offset Value Settings	G10 is not supported.
G17	XY Plane Selection	These commands set the plane in which arcs are executed.
G18	XZ Plane Selection	The setting applies to current and subsequent blocks. The default is G17 (XY).
G19	YZ Plane Selection	
G20	Set Program Units (Inch)	These commands set the unit of measure. The setting applies to current and subsequent blocks. The default is G20 (INCH).
G21	Set Program Units (MM)	
G28	Return to Home Reference	**MILL*PWR*** does not have a method for establishing a "home" position. If one or more coordinates are specified in the block, the table/quill will rapidly move to that location. Program execution will continue with the next program block.
G30		
G40	Cancel Cutter Compensation	
G41	Cutter Compensation (Left)	**MILL*PWR*** does not support automatic cutter compensation. Enabling compensation (G41 or G42) while a diameter (D is not equal to 0) is in effect will generate a run-time error. Enabling compensation with no tool diameter specified (D = 0) has no effect and is ignored.
G42	Cutter Compensation (Right)	

Code	Description	Comment
G43 G44 G49	Tool Length Offset (+) Tool Length Offset (-) Cancel Tool Length Offset	**MILL*PWR*** does not support tool length offsetting. The offset is retrieved from **MILL*PWR*'s** tool library when a tool change is executed. These commands are ignored.
G54 to G59	Work Coordinate System	**MILL*PWR*** does not support presettable work coordinate systems. Selecting a coordinate system is possible, but setting it (G10 or G92) will generate a run-time error.
G61 G64	Set "stop" Path Mode Set "continuous" Path Mode	These commands set the path mode. The setting applies to current and subsequent blocks. The default is G64 (continuous).
G80	Cancel Motion Mode	This command cancels the current motion command mode. Motion is reset to G1. X, Y, and Z coordinates are reset to no motion. I, J, and K coordinates are reset to no offset.
G81 to G89	Canned Cycles	**MILL*PWR*** does not support canned drilling cycles (G81) through G89).
G90 G91	Set Offset Mode (ABS) Set Offset Mode (INC)	These commands set the mode for interpreting coordinates. In ABS mode, coordinates are relative to **MILL*PWR*'s** datum. In INC mode, coordinates are relative to the tool's position after completing the previous move. The setting applies to current and subsequent blocks. The default is G90 (ABS).
G92		G92 is not supported.
G94	Set feed/speed units (/sec)	**MILL*PWR*** does not support "per revolution" units. G93 or G95 will generate a run-time error. G 93 & 95 are not supported.
G*		All other G codes not listed will generate a run-time error.
H	Tool Length Offset (Index)	**MILL*PWR*** does not support H-code tool length offsetting. The offset is retrieved from **MILL*PWR*'s** tool library when a tool change is executed. This command has no effect and is ignored.
I J	X Axis Offset to Arc Center Y Axis Offset to Arc Center	The offset applies to current and subsequent arc blocks (G2 and G3). The default is offset.
K	Z Axis Offset to Arc Center	
N	Line Number	Line numbering is optional and for readability only. **MILL*PWR*** does not make use of this information.
O	Program Number	Program number is optional. **MILL*PWR*** does not make use of this information.
M0	Program Stop	This command stops the program after completion of the block. The cursor moves to the subsequent program block. The current settings remain in effect.
M1	Optional Program Stop	This command is ignored. **MILL*PWR*** does not have a switch for selecting whether to stop on this command.

Code	Description	Comment
M2	Program End	This command stops the program after completing the block. The cursor moves to the beginning of the program. The current settings are reset to default values.
M3 M4 M5	Spindle On (CW) Spindle On (CCW) Spindle Off	If spindle control hardware is present, the spindle is turned on or off automatically. If the hardware is not present, the operator is prompted to turn the spindle on or off and/or to set the speed.
M6	Tool Change	If the tool specified is not the current tool (tools do not match), the operator is prompted to manually change the tool. **MILL*PWR*** retrieves the tool length offset from the tool library. If the system does not have repeatable tooling, the operator must establish the Z-axis datum for the new tool before continuing.
M7 M8 M9	Coolant On (Mist) Coolant On (Flood) Coolant Off	If the AMI hardware is present, the coolant is turned on or off automatically. If the hardware is not present, the operator is prompted to turn the coolant on (mist), on (flood), or off.
M30	Program End w / Pallet Shuttle	**MILL*PWR*** does not support control of a pallet changer. This code has the same effect as M2.
M48 M49	Enable Speed/Feed Override Disable Speed/Feed Override	It is not possible to disable feed rate override on **MILL*PWR***. These commands are ignored.
M60	Program Stop w / Pallet Shuttle	**MILL*PWR*** does not support control of a pallet changer. This code has the same effect as M0.
M*		All other M codes not listed will generate a run-time error.
P	Dwell Time	The dwell time is specified in seconds. The setting applies to current and subsequent dwell blocks (G4). The default is 0.0 sec.
S	Spindle Speed	The spindle speed is set to the specified speed (rpm). If automatic spindle control hardware is present, the spindle speed is set immediately. If not present or not enabled, the user is prompted to set the speed. If the spindle is currently off (M5), it will not be turned on unless accompanied by a spindle direction block (M3 or M4). The setting applies to current and subsequent spindle direction blocks. The default is 0 rpm.
T	Tool Selection	The tool selection represents the number of the tool to use at the next tool change. The selection applies to current and subsequent blocks containing a tool change (M6). The default is no tool selection.
X Y Z	X Axis Coordinate Y Axis Coordinate Z Axis Coordinate	The coordinates represent the destination for the G0, G1, G2, or G3 command currently in effect. They use the current units (G20 and G21) and offset mode (G90 and G91).

Additional G-code Conventions for MILL*PWR*

The following lists some of the expectations and limitations of programs imported into **MILL*PWR***.

- Blocks may contain multiple commands and are executed with the following precedence:

  Messages
  Tool Change
  Spindle Control
  Coolant Control
  Dwell
  Motion
  Stop

  • Because your MILLPWR is a 2-axis control, G-code programs must be created for 2-axis operations. All Z-axis info. must be independent (on a separate line) from X- and Y-axis information. Arcs can only be performed in the X-Y plane. An arc or linear move in the X-Z, or Y-Z plane will generate a **RUN TIME ERROR** during program execution.

- When a Z-axis move is required, the program will pause and you will be prompted to position the Z-axis. Once the Z-axis is in position, pressing **GO** will resume program execution.
- Operator comments should be enclosed in parentheses.
- An operator comment with "MSG" appearing within the text is considered a message. The text following "MSG" (up to 22 characters) is displayed to the operator at run-time. Program execution pauses until the operator acknowledges the message.
- Arcs may only be programmed by endpoint and center. Using endpoint and radius will generate a run-time error.
- Parametric programming (use of variables or algebraic operations) is not supported. The characters (#, [,], <, or >) within a block will generate a run-time error.
- Program delimiters ("%") are ignored. Text following the delimiter is ignored.
- Text following a programming comment character (";") is ignored.
- White space is ignored between parameters but not within a numeric value or message.
- If a coolant command (M7, M8 and M9) appears in the block, the Operator Intervention Message is displayed regardless of the current coolant setting. If AMI hardware is present, the block will execute without the need for any operator intervention or acknowledgement.
- All tool diameter compensation must be made in CAD/CAM, however, the tool length offset is read from **MILL*PWR***'s tool library.

  For example: In a G-code file, T1 will use the tool length offset from Tool #1 in the tool library. T2 will use tool length offset from tool library tool #2, etc.

- The skew feature does not work with G-code programs. Remove any skew angle prior to running a G-code program.

Merging Programs

The **MERGE** softkey allows you to merge two **MILL***PWR* programs together. With this function, all of the program steps within the program you selected will be copied into your current or opened program. Keep in mind that you can edit any of the new steps at a later time if you need to (refer to **Step Functions**).

It's always a good idea to save your work before merging programs so that you can easily recover your original program if you need to.

To merge programs:

• From the PGM screen, highlight the step in your current (open) program where you want the new steps to be inserted.

• Press the **PROGRAM FUNCTIONS** softkey.

• Press the **MERGE** softkey. The last directory that was opened will appear. You can change directories if you need to (refer to **Selecting a Directory**).

• Using the arrow keys, highlight the program that you want to merge into your program.

• Press the **MERGE** softkey again. The steps from the program you highlighted should now appear in your program.

Note: The merge function does not work with G-code files or DXF files.

Backing Up a Program

With the **BACKUP** softkey, you can make backup copies of programs that you have already saved on **MILL*PWR***'s internal memory. You should keep backup copies on hand in case a program is accidentally deleted or modified, or you can't recover the original programs for any other reason.

It's best to save the final version of a program before creating a backup copy. Otherwise, you'll have to back up the program again after you've made any changes.

To back up a program:

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then select the directory containing the program(s) you want to back up (refer to **Selecting a Directory**).

- Now press the **BACKUP** softkey. The following softkey options should appear:

SELECT A BACKUP FUNCTION.					
ALL PROGRAMS	SELECT PROGRAMS			USE FLOPPY	REMOTE STORAGE

- Indicate where you want to back up your program(s): onto a $3^1/_2$" floppy disk (press the **USE FLOPPY** softkey) or onto your PC (press the **REMOTE STORAGE** softkey).

- Determine which program(s) you want to back up.

    To back up all of your programs in the directory:

        Press the **ALL PROGRAMS** softkey. **MILL*PWR*** will highlight and store a backup copy of each program in the directory you have selected.

    To back up only one or just a few programs:

- Press the **SELECT PROGRAMS** softkey.

- Highlight each program you want to back up and press the **SELECT PROGRAM** softkey again. An arrow will appear beside each program name you have selected.

- Now press the **BACKUP PROGRAMS** softkey. **MILL*PWR*** will highlight the program(s) and save a backup copy in the directory you've selected.

Note: If a program with the same name is already stored in the directory you've chosen, **MILL*PWR*** will ask you if you want to replace the old copy with the latest copy. Choose the **YES** softkey or "**yes**" to ALL soft key to continue or the **NO** softkey to cancel.
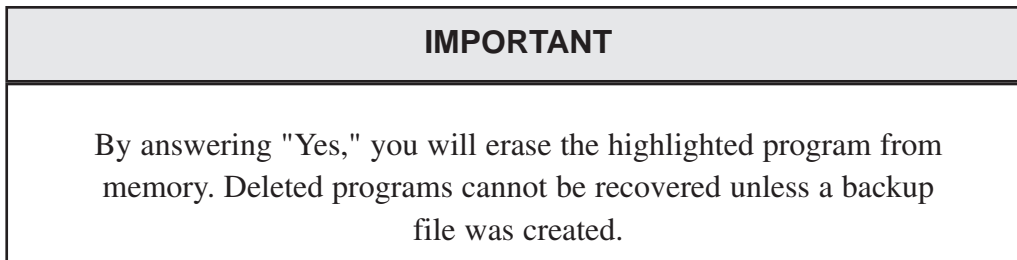
Deleting a Program

You can delete any program that has been saved.

To delete a program:

- Select the directory that contains the program you want to delete (refer **Selecting a Directory**).

- From the PGM screen, press the **PROGRAM FUNCTIONS** softkey, then press the **DELETE** softkey.

| SELECT A PROGRAM FUNCTION.					
LOAD	SAVE	DELETE	MERGE	BACKUP	DIRECTORY

- Using the arrow keys, highlight the program you want to delete.

- Press the **DELETE** softkey. You will be asked if you are sure that you want to delete the program. Press the **YES** softkey to continue or the **NO** softkey to cancel.

IMPORTANT
By answering "Yes," you will erase the highlighted program from memory. Deleted programs cannot be recovered unless a backup file was created.

This page intentionally left blank.

Demonstration Program

The following steps and key stroke sequences will guide you through creating a demonstration program based upon information from the print below. This will help introduce you to **MILL*PWR*** and familiarize you with how it operates.
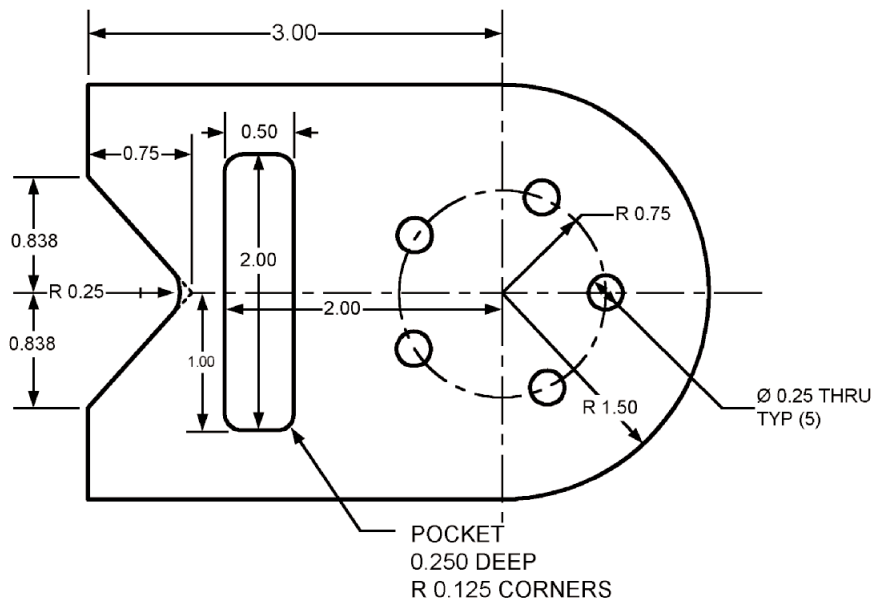
Selecting Datum

Although there is no clear "zero point" identified on this print, use the center of the bolthole pattern as the datum. There are several advantages to using this location:

• It is the center of the bolthole pattern.

• It is the center of the large arc, making it easy to calculate the arc's start and end points.

• The pocket is dimensioned from this point.

• Nearly all but one of the dimensions on the print originate from this point.

This print could also be programmed using another point—the upper left corner, for example—as datum.

Set datum when you are ready to run the program.

Beginning The Program

Begin by pressing the **PGM** key.
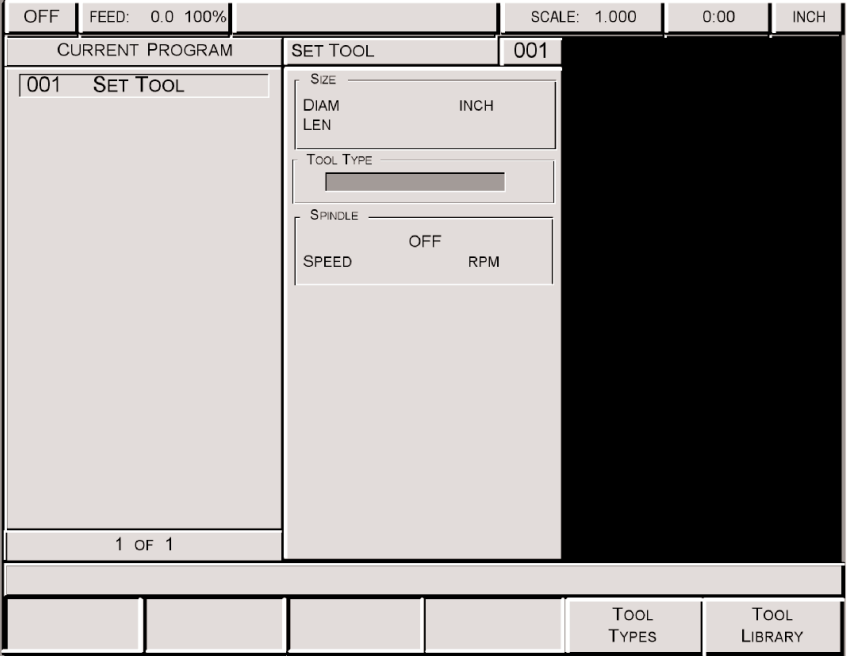
Selecting A Tool

The logical first step for most programs is to choose the tool
that you want to begin with. Let's use a 1/4" **FLAT END MILL**.

- Press the **TOOL** key.

- Enter a value of 0.25.
  The tool length is optional— leave it blank.

- Arrow down to **TOOL TYPE.**

- Press the **TOOL TYPES** softkey.

- Arrow down to **FLAT END MILL** and press the **ENTER** key.

Consider entering a Position/Drill step before the Set Tool step. This will enable you to go to a location away from the work-piece to change tools.

*Also, leave the diameter blank and MILL*PWR* will ask for the tool diameter when you run the program.*

| OFF | FEED: 0.0 100% | | | SCALE: 1.000 | 0:00 | INCH |

```
       CURRENT PROGRAM          SET TOOL           001
  001    SET TOOL                ┌ SIZE ─────────────────────┐
                                 │ DIAM            INCH       │
                                 │ LEN                        │
                                 └────────────────────────────┘
                                 ┌ TOOL TYPE ────────────────┐
                                 │ ▭                          │
                                 └────────────────────────────┘
                                 ┌ SPINDLE ──────────────────┐
                                 │            OFF             │
                                 │ SPEED           RPM        │
                                 └────────────────────────────┘


            1 OF 1
```

| | | | | TOOL TYPES | TOOL LIBRARY |

- Arrow down to **SPINDLE**.

- To use a forward spindle direction.
  Press the **FORWARD** softkey then
  press the **ENTER** key.

- Set the cutter's spindle speed for 1300 RPM.

- Press the **USE** key.

You don't have to press the **ENTER** key after the last
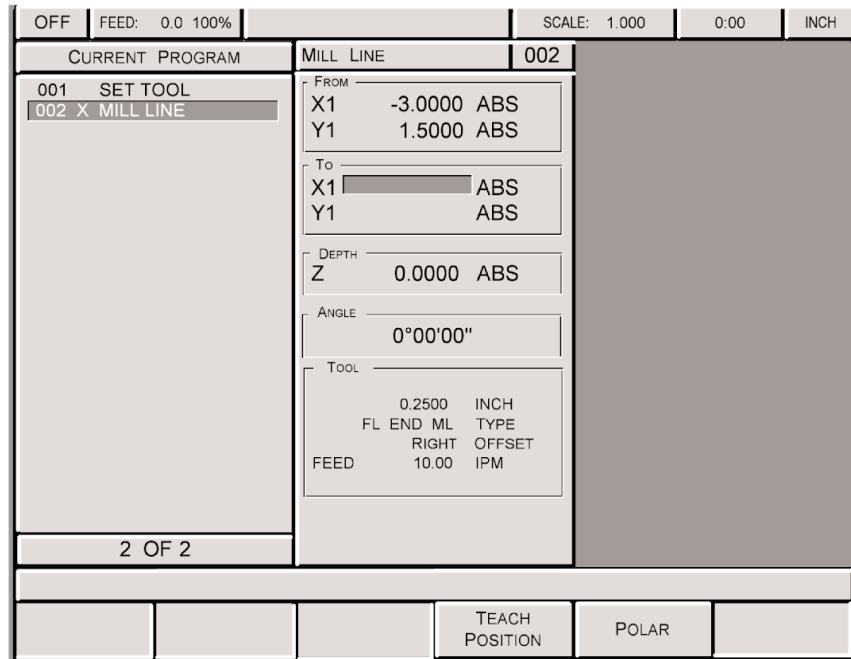value—you can just press **USE**.

Programming the Contour

This part can begin at several different places. Begin at the upper-left corner, and cut in a clockwise (CW) direction.

•   Press the **LINE** key.

| OFF | FEED: 0.0 100% | | SCALE: 1.000 | 0:00 | INCH |

CURRENT PROGRAM | MILL LINE | 002

001   SET TOOL
002 X MILL LINE

To enter a negative number, use the "±" key, not the "-" or "¥"key. The "-" and "¥"keys are for performing math operations within a numeric field.

FROM
X1   -3.0000 ABS
Y1   1.5000 ABS

TO
X1   ABS
Y1   ABS

DEPTH
Z   0.0000 ABS

ANGLE
0°00'00"

TOOL
0.2500   INCH
FL END ML   TYPE
RIGHT   OFFSET
FEED   10.00   IPM

2 OF 2

TEACH POSITION | POLAR

•   Using the keypad, enter the following information:

        FROM: X1 = -3 ABS

                  Y1 = 1.5 ABS

        TO: X2 = 0 ABS

                  Y2 = 1.5 ABS

        DEPTH: Z = -.25 ABS

- Now highlight the "Offset" field.

The tool specifications will be filled in automatically
from the information we entered in step 001.

- To cut around the outside of the contour in a clockwise
  direction, use a left offset.

- Press the **LEFT** softkey.

LEFT

MILL*PWR* has been factory set with a feed rate of
10 inches per minute, which is fine for this operation.

- Press the **USE** key.

USE

Notice that the line is immediately displayed on the Program (PGM)
screen. This is called immediate-part-view-graphics.

Next program the arc:

- Press the **ARC** key.

MILL*PWR* assumes that you are continuing from where you left off so it automatically fills in the "From" point, the depth and tool information for you.

- Enter the following information into the "To" and "Radius" fields:

> **TO**:   X2 = 0 ABS
>
>   Y2 = -1.5 ABS
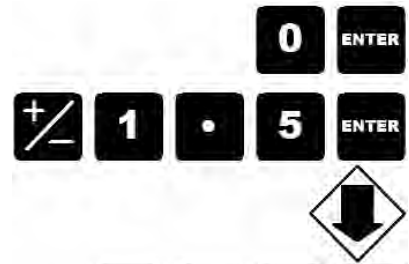
> **RADIUS**:   1.5 MINOR

Since this arc will be starting at the top and moving around to the bottom, choose **CW** for the direction.

> **DIRECTION:**   CW

- Press the **USE** key.

OFF | FEED: 0.0 100% | TOOL: 0.1250 INCH FL END ML | SCALE: 1.0000 | 0:00:00 | INCH

CURRENT PROGRAM

001   SET TOOL
002 ⌐ MILL LINE
003 L MILL ARC

END OF 3

PROGRAM FUNCTIONS | CLEAR PROGRAM | STEP FUNCTIONS | MORE STEPS | | RUN OPTIONS
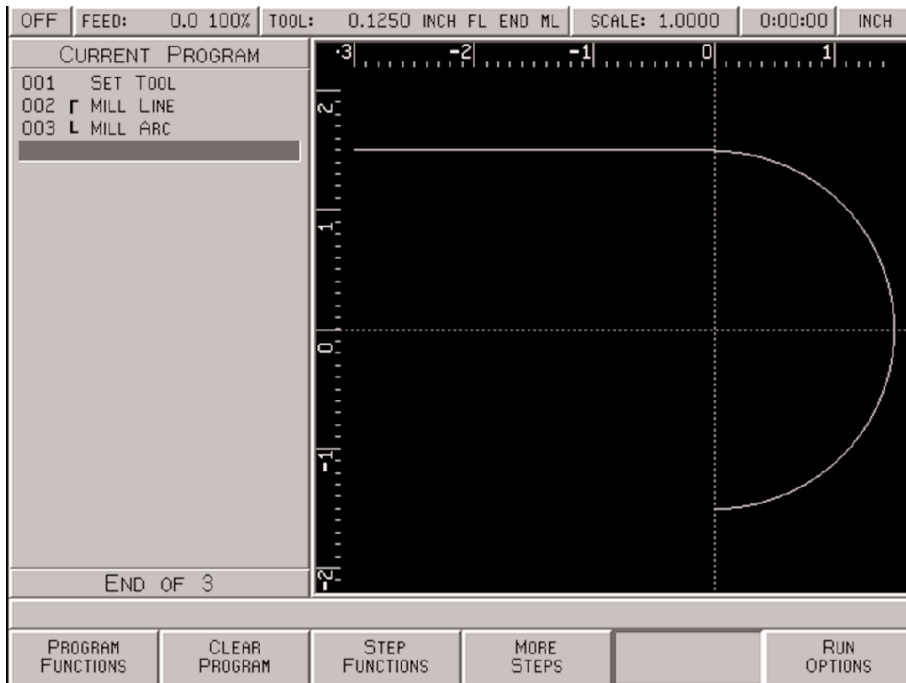
Notice the lines connecting steps 002 and 003 in the program list. This indicates that the line and arc form a continuous con-tour. MILLPWR will cut them without stopping.

Next, enter the line that starts at the bottom of the arc.

- Press the **LINE** key and enter the following information:

  TO: X2 = -3.0 ABS

  Y2 = -1.5 ABS
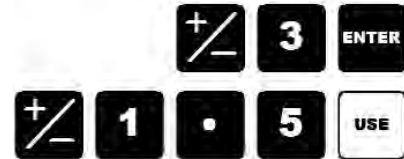
- Press the **USE** key.

- Press the **LINE** key to add another line. Now enter the following information:

  TO: X2 = -3.0 ABS

  Y2 = -0.838 ABS

- Press the **USE** key.

```
OFF  FEED:    0.0 100%  TOOL:    0.1250 INCH FL END ML   SCALE: 1.0000   0:00:00   INCH

          DEMO
001   SET TOOL
002 ┌ MILL LINE
003 │ MILL ARC
004 │ MILL LINE
005 └ MILL LINE




          END OF 5

PROGRAM      CLEAR       STEP        MORE                    RUN
FUNCTIONS   PROGRAM   FUNCTIONS    STEPS                  OPTIONS
```
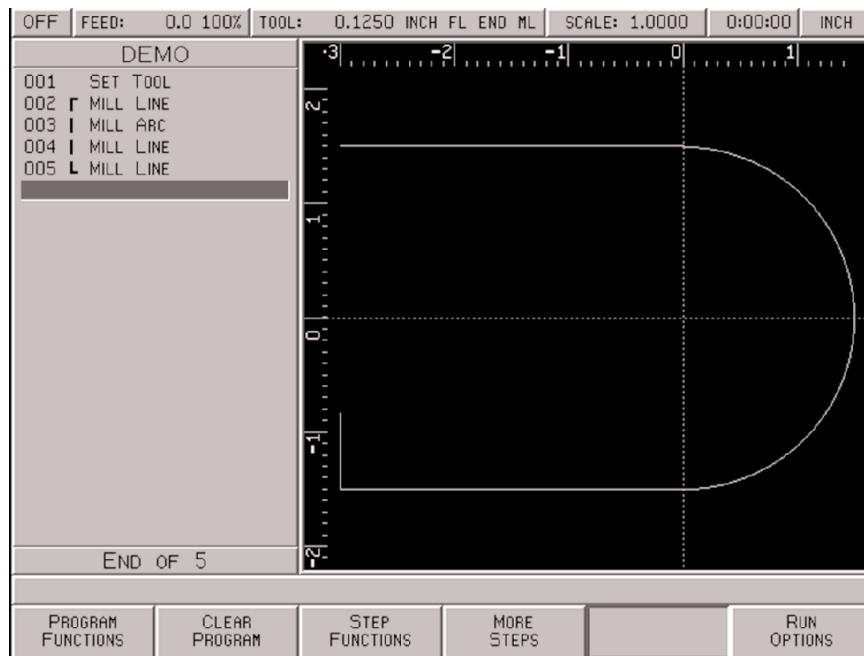
- Press the **LINE** key.

- Enter the following information:

    TO: X2 = .75

- Now press the **ABS/INCR** key.

MILL*PWR* assumes that you want to use the
X coordinate of the "From" point (X1) as your
incremental reference, which is exactly what
is needed.

- Press **ENTER** to confirm this and again to accept the
  value.

- Now enter the location of Y2 in the "To" field:

    TO: Y2 = 0

- Press the **USE** key.

| OFF | FEED: | 0.0 100% | TOOL: | 0.1250 INCH FL END ML | SCALE: 1.0000 | 0:00:00 | INCH |

```
       DEMO
001   SET TOOL
002 ┌ MILL LINE
003 │ MILL ARC
004 │ MILL LINE
005 │ MILL LINE
006 └ MILL LINE



        END OF 6
```

MILL LINE   006

FROM
X1   -3.0000 ABS
Y1   -0.8380 ABS

TO
X2   0.7500 INC X1
Y2   ABS

DEPTH
Z   0.2500 ABS

ANGLE
       0°00'00"

TOOL
       0.2500 INCH
  FL END ML TYPE
       LEFT OFFSET
FEED   10 IPM

| PROGRAM FUNCTIONS | CLEAR PROGRAM | STEP FUNCTIONS | MORE STEPS | | RUN OPTIONS |

- Press the **LINE** key again.

    TO: X2 = -3.000 ABS

            Y2 = 0.838 ABS

- Press the **USE** key.


```
OFF  FEED:    0.0 100%  TOOL:    0.1250 INCH FL END ML   SCALE: 1.0000    0:00:00    INCH
                DEMO              -3 . . . . . . -2 . . . . . . -1 . . . . . . 0 . . . . . . 1 . . .
001    SET TOOL
002 ┌ MILL LINE
003 │ MILL ARC
004 │ MILL LINE
005 │ MILL LINE
006 │ MILL LINE
007 L MILL LINE




            END OF 7

  PROGRAM      CLEAR       STEP        MORE                    RUN
 FUNCTIONS    PROGRAM    FUNCTIONS    STEPS                  OPTIONS
```
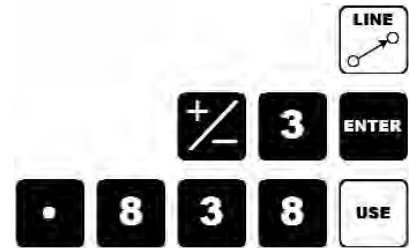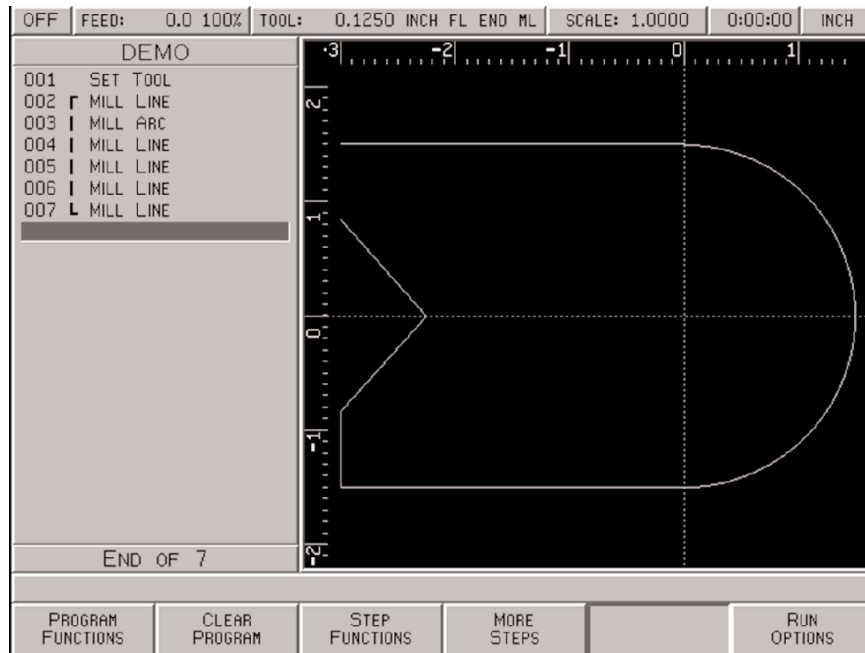
Now go back and insert a **BLEND** between steps 006 and 007.

- Using the **UP** arrow key, highlight
  the last step, 007 Mill Line.

- Press the **BLEND** key.

You can see in the program list that step 007 has moved
to step 008 and that the blend will be inserted into step 007.

Notice the "Steps" field informs us that steps 6 and 8 will
be blended.

- Enter a radius of 0.25 and press the **USE** key.
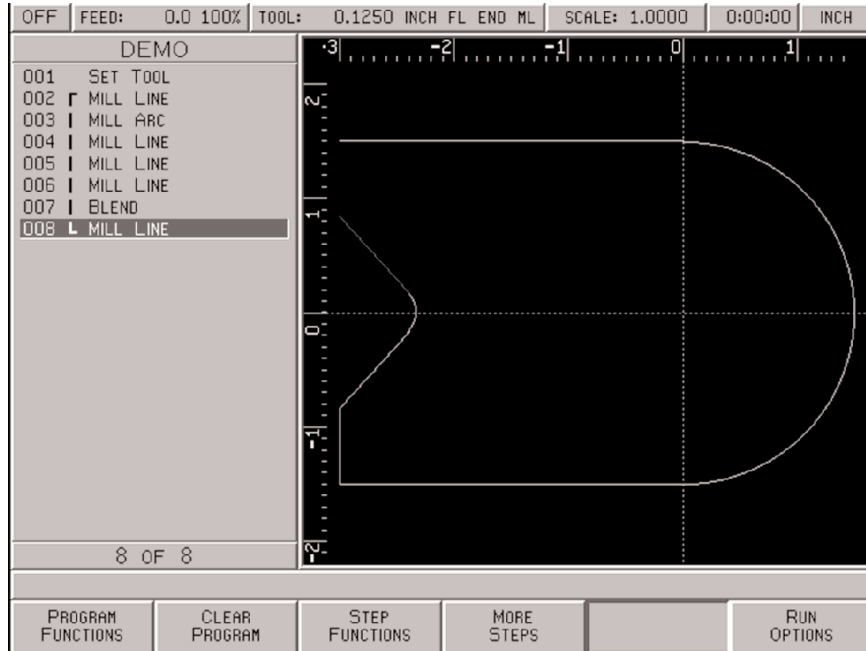
Notice how the last two lines are now "blended" together with a radius.

The **BLEND** step could just as easily have been inserted immediately after line 006. In doing so, the **BLEND** step *would* show up in the listing, but it *would not* be displayed graphically until the step that follows is added.

- Press the down arrow key once to reach the end of the program. Now more steps can be entered.

- Press the **LINE** key.

  TO: X2 = -3.0 ABS

  Y2 = 1.5 ABS

- Press the **USE** key.

Notice how the lines next to the program steps have changed. They now indicate that the contour is closed.

This happens when the "To" point of the last step is the same as the "From" point of the first step.

```
CURRENT PROGRAM

001     SET TOOL
002 Π MILL LINE
003 II MILL ARC
004 II MILL LINE
005 II MILL LINE
006 II MILL LINE
007 II BLEND
008 II MILL LINE
009 U MILL LINE
```
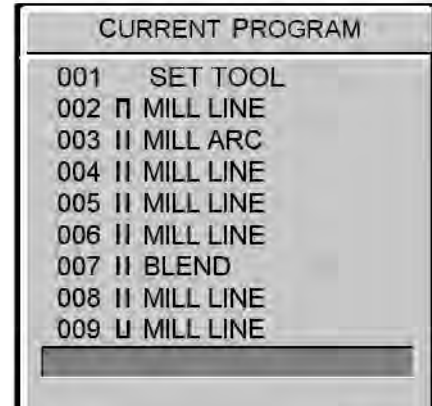
Programming the Bolthole Pattern

Begin by changing the tool.

- Press the **TOOL** key and enter 0.250" for the diameter.

  Again, skip the tool length.

- In the **TOOL TYPE** field, arrow down to **DRILL** and press **ENTER**.

- Adjust the spindle direction and speed as needed.

- Press the **USE** key.
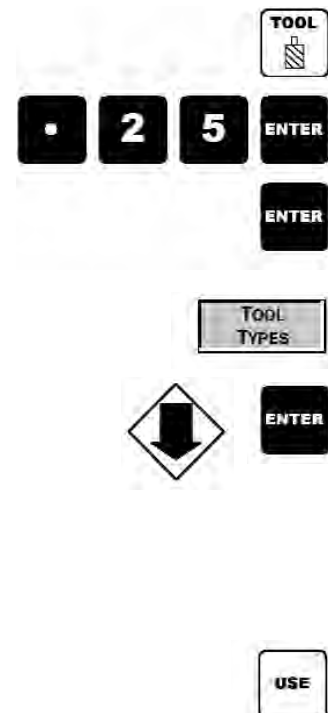
• Press the **HOLES** key and then press the **BOLT CIRCLE** softkey.

Notice that the **CENTER** and **DIRECTION** are carried forward from the arc that we programmed earlier. Leave the depth of the holes alone and change the radius and enter the number of holes.

• Arrow down to the "Radius" field and enter a value of 0.75".
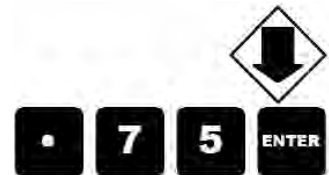
• Press the **CCW** softkey for a counter-clockwise direction.

• Enter a value of 5 for the number of holes.

• Press the **USE** key.

In this example, the starting and ending angles were not changed. As a result, the first hole will be placed at zero degrees. If you were to look at the face of a clock, zero degrees will be at 3 o'clock. Without a specified ending angle **MILL*PWR*** will space the number of holes entered evenly around a full circle. If a counter-clockwise direction was applied to this hole pattern, the second hole of the five hole pattern will be between 12 o'clock and 1 o'clock. The holes continue around the circle as shown below.
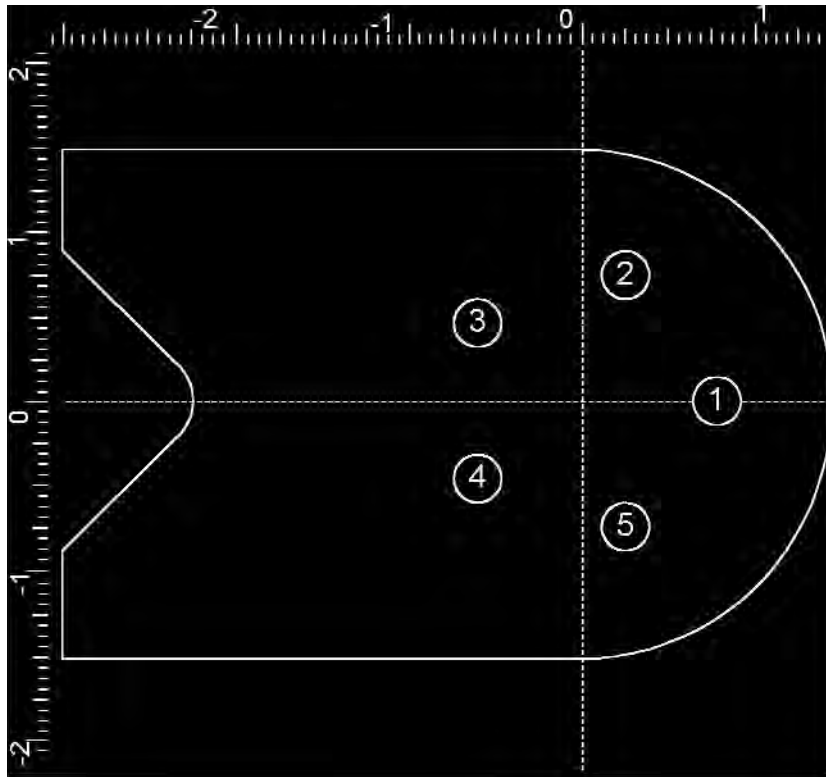
Programming the Rectangular Pocket

First, enter the tool that is needed to
machine the pocket.

- Press the **TOOL** key and enter the data for a
  0.125" diameter flat end mill.

- Adjust the spindle direction and speed if you want.

- Press the **USE** key.

- Press the **RECT** (rectangle) key, then press the
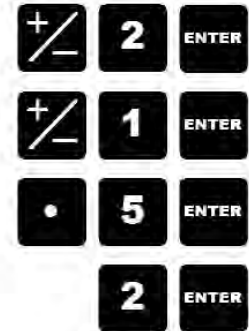 **POCKET** softkey.
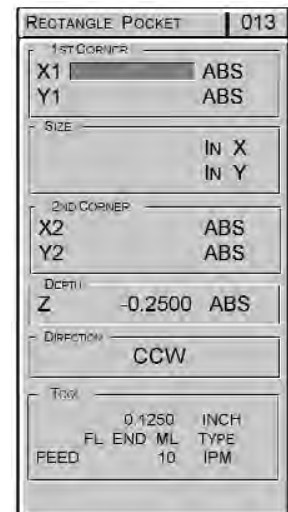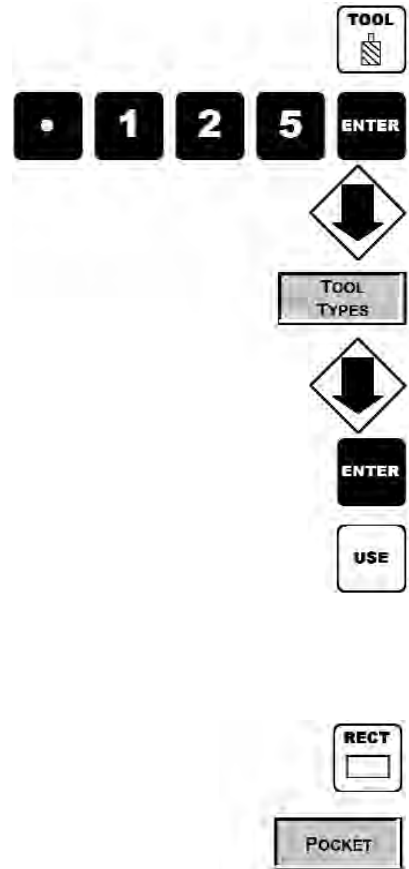
• Enter the following information:

1ST Corner: X = -2.0 ABS

                      Y = -1.0 ABS

SIZE: 0.5 in X

                      2.0 in Y

DIRECTION: CCW

• Press the **MORE** softkey.

CORNER BLEND RADIUS: .125

• Press the **USE** key.

Saving Your Program

Your demonstration part program is now complete.

- To save the program, press the **PROGRAM FUNCTIONS** softkey.

- Press the **SAVE** softkey.

- You can name the program by pressing the numeric keys or by pressing the **ALPHABET** softkey, highlighting a letter, and then pressing the **ENTER** key. You may select up to eight characters, mixing numbers and letters if you wish.

- After naming the program, press the **SAVE** softkey to save it.

You will be warned if a program already exists with the name that you have entered.

Saving the program means that it is stored and will not be lost if there is a power interruption.

You can retrieve the program later by pressing the **PROGRAM FUNCTIONS** and **LOAD** softkeys.

Testing Your Program

It's always a good idea to test the program before you cut a part.

- Press **RUN OPTIONS** and then select both the **DRY RUN** and **GRAPHICS ONLY** softkeys.

- Move to the start of the program by pressing the **1** key followed by the **ENTER** key.

- Now press the **GO** key. Watch as **MILL*PWR*** shows how the part will be cut.

- Before attempting to cut the part, press the **RUN OPTIONS** softkey and make sure the **DRY RUN** and **GRAPHICS ONLY** keys are not selected.
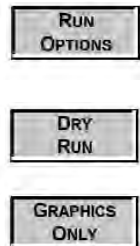
- Now press the **DRY RUN** and **GRAPHICS ONLY** softkeys to de-select them.

Running the Program

The first step in running a new program is to establish datum.
Remember that to chose the center of the bolt circle as datum.

- Place the workpiece into a vise.

- Position the tool over the workpiece where you
  want the center of the bolt circle's datum to be.

- From the DRO screen, press the **DATUM** softkey.

- Press the **X = 0** and **Y = 0** softkeys to establish the
  current tool position as datum, and then press the **USE** key.
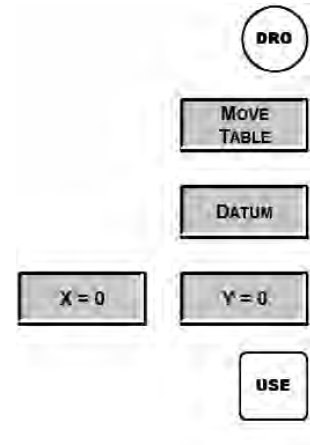
- Return to the program by pressing the **PGM** key.

Perform a second dry run, this time allowing the table to move
to see if we have correctly set Zero.

- Press **RUN OPTIONS** then press **DRY RUN**.
  Check that **GRAPHICS ONLY** is *not* selected.
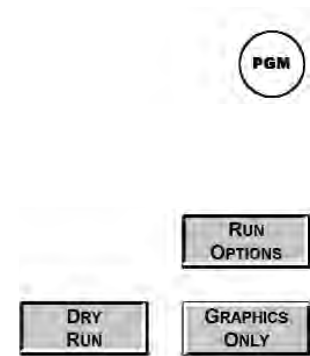
*If the cursor
is positioned
past the last
step,* **MILL*PWR***
*will automati-
cally go to step
001 when you
press GO.*

- Now press the **GO** key and check that the tool is
  clear of any obstruction.

- Press the **GO** key again. **MILL*PWR*** will run each
  step without stopping.

Tool Changes

Whenever you encounter a **SET TOOL** step, **MILL*PWR*** will display the DRO screen and let you know which tool to load.

- Use the **MOVE TABLE** softkey and arrow keys to move the tool away from the workpiece.

- Insert the required tool into the spindle.

- Using the **MOVE TABLE** feature, position the tool over the surface of a known depth on the workpiece.

- Press the **DATUM** softkey.

If the surface is at 0, use the Z = 0 softkey.

- Position the tool so that it touches the surface of the workpiece, then enter that position into the Z-axis.

- Press the **USE** key.

- Raise the tool and press the **GO** key to run the demonstration program.

Clearing the Program

- From the PGM screen, press the **CLEAR** softkey to clear the program from the screen.

- Press the **YES** softkey. The PGM screen will be cleared and ready for another program.

Simple Milling & Drilling

Set Tool

"Set Tool" defines the tool and should appear as the first step in all of your programs. You should also insert a "Set Tool" step anywhere you want to change tools. **MILL*PWR*** will apply the last tool setting to the program steps that follow.

To program a "Set Tool" step:

• Press the **TOOL** key.

• Enter the tool's diameter.

> *Note:* If you leave the "Diam" field blank, you will be prompted to provide a diameter once the program is running.

Consider entering a Position/Drill step before the Set Tool step. This will enable you to go to a location away from the workpiece to change tools.

> *Note:* Tool length offsets are mentioned later on within this section.

   • Highlight the "Tool Type" field.

   • Press the **TOOL TYPES** softkey for a list of available tool types.

• Highlight a tool type, then press the **ENTER** key.

• For spindle control, select from the **OFF**, **FORWARD** and **REVERSE** softkeys.

   If you selected "Forward" or "Reverse," enter a spindle speed. An Operator Intervention Message (OIM) will prompt you to set the speed and direction when you run a program. (If your machine's set up to control the spindle, **MILL*PWR*** will adjust the speed and direction for you.)

• Press the **USE** key.

Programming a Tool Step with Repeatable Tool Length Offsets

If you begin setting the tool length offsets by setting Datum using an electronic edge finder it must have a fixed and repeatable length. The current tool information cannot have a length value. All of the tool length offsets in the program will be the difference in length between the tool and the edge finder.
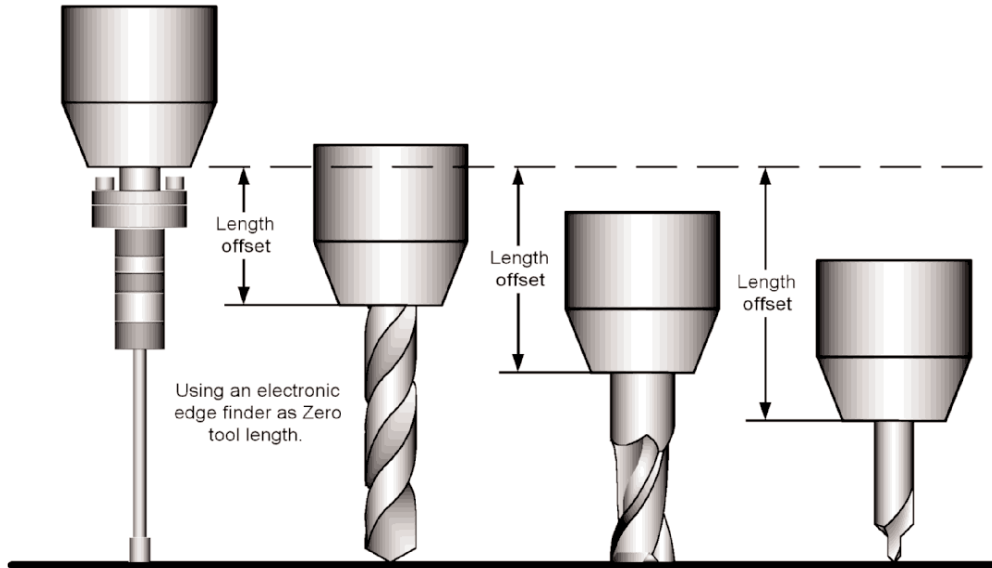
When defining tool length offsets in the Tool Step, touch each tool to the same surface and press the **TEACH TOOL LENGTH** softkey to establish the tool length.

Once you have identified the tools you will be using, you can program the length offsets into the tool steps.

Entering the first tool

Set the Z-axis Datum with a tool or standard using no tool length offset. In this example, the first tool in the program is used to set datum zero. If you wish to use an electronic edge finder or tool holder with a repeatable standard, then place it into the spindle instead, and set datum. Then you find the tool length offset for tool 1 as well.

- From the **DRO** screen, press the **TOOL** key.
- Enter the diameter and then enter zero for the tool length.
- Press **GO**.
- An Operator intervention message will appear asking you to "use" the entered tool.
- Place the first tool in the spindle and press **GO** again.
- **MILLPWR** is now set with a zero tool length offset.
- Press the **DATUM** softkey.
- Touch the current tool to the top of the workpiece.
- Press the "**Z=0**" softkey, then the **USE** key.

Use this same location to "teach" each tool length in the program.

- Press the **PROGRAM** key and highlight the first tool step.
- Press **ENTER**.
- Use the numeric keypad to enter the tool's diameter.
- Highlight the length field and ensure the length offset is zero. (If you used an electronic edge finder or tool holder with a repeatable standard to set datum, then touch tool 1 to the same surface you used to set the Z-axis datum above and teach its tool length here.)
- Highlight the next tool step and press **ENTER**.
- Use the numeric keypad to enter the tool's diameter. If the diameter you entered is metric, press the **MM** softkey.
- Place this tool into the spindle and touch its tip to the same surface you used to set the Z-axis datum Zero above.
- Press the **TEACH TOOL LENGTH** softkey. The length offset for this tool will appear in the length field.
- Highlight the "Type" field.
- Press the **TOOL TYPES** softkey.
- Highlight the type of tool from the list of available tool types and press **ENTER**.
- Press **USE**, and repeat this procedure for each additional tool in the program.
- If the Z-axis datum is not located at the top of the workpiece, you will need to reset the Z-axis datum. Be sure to set the Z-axis datum using the tool shown in the "current tool" box located in the status bar on the top of the screen. **If the tool doesn't match, the datum will not be correct.**

After the tool information is established in the program, later you can edit the values as needed to adjust for accuracy and wear.

Changing to a Tool of Unknown Length When in the DRO

If you are using the DRO and need to set a new tool, follow the procedure below. Be sure to reestablish the tool length offset for theprogram before running it as described below.

- From the **DRO** view, press the **TOOL** key.
- Enter the tool diameter, clear any length and if you wish, the type.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that you have installed the tool.
- Set the Z-axis datum following the instructions found in the operation manual.

At this point, the program's tool length offsets are no longer valid. Follow the steps below to reestablish the Z-axis datum and tool length offsets for the program.

- From the **PROGRAM** view, highlight any tool step.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the programmed tool.
- Place this tool in the spindle.
- Press the **DATUM** softkey.
- Reset the Z-axis datum following the instructions found in the operation manual.
  By resetting Datum during a tool step within the program, all of the tool offsets in the program become valid again.

Changing to a Tool of Unknown Non-Repeatable Length when Executing a Program

When running a program, it is easy to switch between tools of known length, to tools that have a non-repeatable length such as a counter-bore held in an R-8 collet. For example, let's say a program is being run that has tool 1 -a center drill- in a repeatable holder, as the first step.

Step 2 in the program is a bolthole pattern.
Step 3 is a tool step using tool two-a drill- in a chuck that also repeats.
Step 4 is the same hole pattern repeated.
Step 5 is a tool step programmed as a counter bore of unknown length.
Step 6 is the same hole pattern repeated.

For this example, if two identical parts are being machined. The first part is in place and datum has been set using tool one from the program as the current tool.

The top of the workpiece is Z-zero. When the **GO** button is pressed, **MILL*PWR*** will see that step one is asking for the same tool as the current tool and will proceed on the next step. When the next tool step is executed, you will be prompted to change tools to the drill. Simply change the tool and press **GO** to continue. The next tool-Step 5- is of unknown length. When this step is executed, and you are prompted to change tools,

- Press the **DATUM** softkey. The **DRO** screen will appear with the set datum window.
- Place the counter bore in the spindle and touch the cutting edge to the top of the workpiece.
- Press the **Z=0** softkey and then **ENTER**. The tip of the tool is now set at datum zero.
- Press **GO** to continue with the program.

Since the Z-axis datum was set using a tool of unknown length, the programmed tool lengths will not repeat until the Z-axis datum is reset again, using a tool of known length. This can be done when step 1 is run on the second part. After the part has been changed,

- Press **GO**. You will be prompted to change tools to the center drill.
- Press the **DATUM** softkey. The **DRO** screen will appear with the set datum window.
- Place the center drill in the spindle and touch the tip to the top of the workpiece.
- Press the **Z=0** softkey and then **ENTER**. The tip of this tool is now set at datum zero.
- Press **GO** to continue with the program.

Now that datum has been reset with a tool of known length, all of the preset tools again relate to datum. The program can continue as usual.
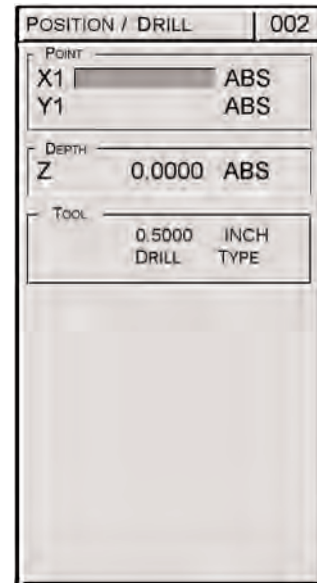
Position/Drill

POS
→○

The position/drill function will move the table to the position you want based upon your X- and Y-axes coordinates.

To program a position/drill step:

- Press the **POS** key.

- Enter the X- and Y-axes coordinates.

- Enter a depth.

- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

```
POSITION / DRILL      | 002
┌ POINT ──────────────────
 X1 ▓▓▓▓▓▓▓▓▓    ABS
 Y1              ABS
┌ DEPTH ──────────────────
 Z        0.0000 ABS
┌ TOOL ───────────────────
          0.5000 INCH
          DRILL  TYPE
```

Center Line

With the position/drill feature, you can also locate the midpoint of two points or the center line of a circle.

If you use an electronic edge finder, the points will automatically be entered on contacteven if you over-travel.

To locate a center line:

- Press the **CENTER LINE** softkey.

- Locate the first edge and press the **ENTER** key.

- Locate the second edge of your part and press the **ENTER** key.

- If you are calculating the center of a circle, locate and enter a third point along the diameter of the circle.

- Press the **USE** key.

Line

Lines are defined by their "From" point (the point where they begin) and "To" point (the point where they end).

There are two ways you can program a line:

- With four coordinates (X1, Y1, X2, Y2)

- With three of the coordinates above (X1, X2, Y2 or X1, Y1, X2, etc.) and an angle

Keep in mind that the angle is measured from the line on the X-axis and sweeps towards the adjacent line.

Choose a method based upon the information available from the print.

To program a line:

- Press the **LINE** key.

- Enter the beginning X- and/or Y-axes coordinates into the "From" field.

- Enter the ending X- and/or Y-axes coordinates into the "To" field.

- Enter a depth.

- If one of the X- or Y-axes fields above was left blank, enter an angle.

- Highlight the "Offset" field and press the **LEFT**, **CENTER,** or **RIGHT** softkey.

- Enter the feed rate.

- Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).
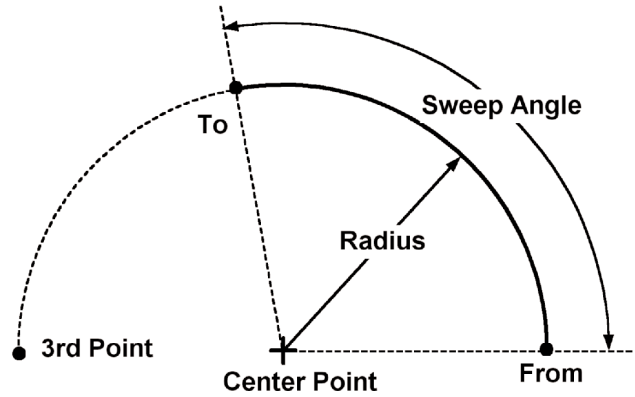
Arc

An arc can be defined several ways:

- With a from point, to point and a radius

- With a from point, to point and a center point

- With from, to, and 3rd points along the arc

- With a from or to point, center point and a sweep angle.

Choose a method based upon the information available from the print.

As you are programming, keep in mind that the arc's sweep angle is measured from the X-axis.

To program an arc:

- Press the **ARC** key.

- Enter the beginning coordinates for the X- and Y-axes in the "From" field.

- Enter the ending coordinates for the X- and Y-axes in the "To" field.

- Enter a depth.

- Enter the arc's radius, then press either the **MINOR ARC** or **MAJOR ARC** softkey. (A major arc has a sweep angle greater than 180 degrees; a minor arc's sweep angle is less than 180 degrees.)
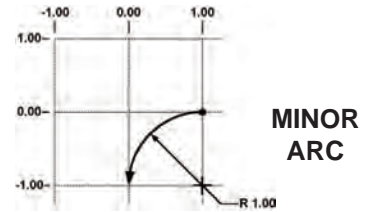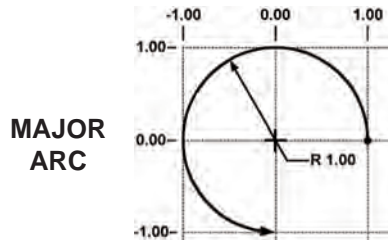
Both arcs have the same starting point, ending point and radius.

- Select the cutting direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

- Highlight the "Offset" field and press the **LEFT**, **CENTER**, **RIGHT, INSIDE** or **OUTSIDE** softkey.

- Enter a value for the feed rate.

- If you need to enter a center coordinate, 3RD point or sweep angle, press the **MORE** softkey. Otherwise, press the **USE** key.

Center
    Enter the center coordinate's position for the X- and Y-axes.

3RD Point
    Enter your 3RD coordinate's position for the X- (X3) and Y-axes (Y3).
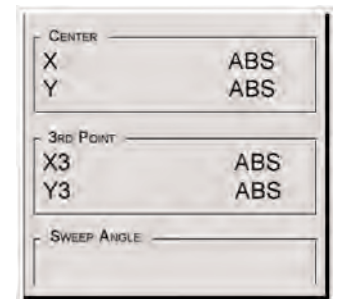
Sweep Angle
    Enter the sweep angle.

Information that appears in blue has been calculated. If any of these values are already displayed in blue, then **MILL*PWR*** has enough data for the arc and has calculated the rest.

Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Blend

A blend is an arc that connects two lines, two arcs or a line and an arc. All you have to do is provide the radius for the blend and indicate whether it is normal or inverted. **MILL*PWR*** will calculate the tangent points for you.

The two steps you want to blend can, but don't have to, intersect or touch. If they don't come into contact with each other, check that the radius is large enough to connect them.
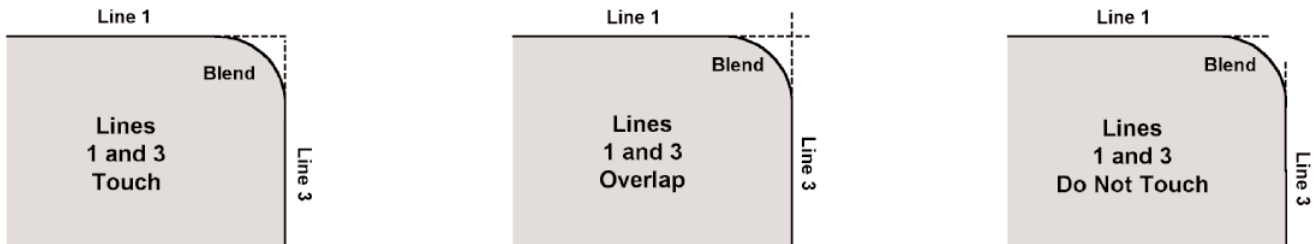
It's also possible to close a contour (e.g., a triangle) using the blend feature by inserting a blend step immediately after the last step in the contour.

After you enter the blend's radius, press the **CLOSE CONTOUR** softkey, and **MILL*PWR*** will blend the last step with the first step.

To program a blend:

- Highlight a step within the program where you want to place a blend.

- Press the **BLEND** key.

- Check that the steps listed in the "From" and "To" fields are the steps you want to blend. (If they are incorrect, press the **CANCEL** key and highlight the appropriate step.)

- Enter the blend's radius. (Press the **CLOSE CONTOUR** softkey if you want to blend the end of a contour with the beginning. The step numbers in the "To" and "From" fields will automatically change.)

- Press either the **NORMAL ARC** or **INVERTED ARC** softkey. A normal arc curves outward; an inverted arc curves inward.
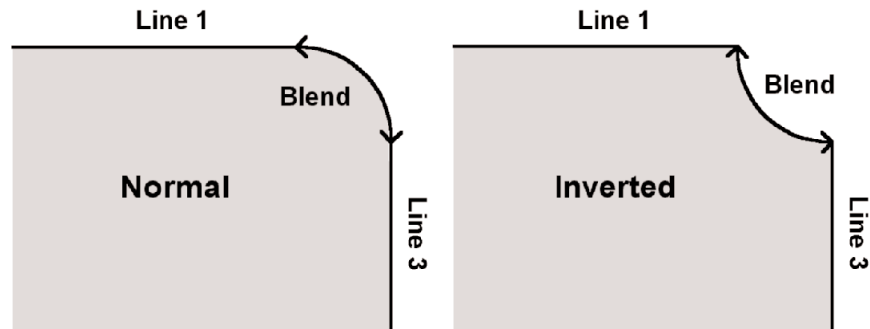
- Enter the feed rate.

- Press the **USE** key.

## Rectangular Milling Functions

MILL*PWR* offers several rectangular milling functions that let you program pockets, frames, faces and slots quickly and easily.

POCKET	FRAME	FACE	SLOT		

Pocket

A pocket is a cavity or area on a part where material is removed when you machine. You can program a rectangular pocket two ways:

- Using the coordinates of two diagonal corners.

- Using the coordinates of one corner and the size of the pocket.
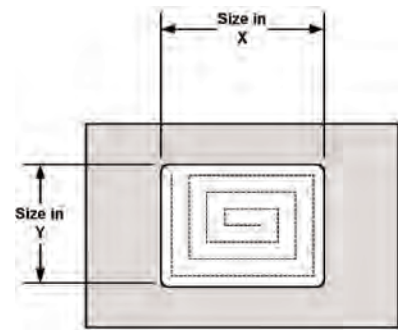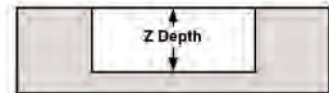
To program a rectangular pocket:

- Press the **RECT** key.

- Press the **POCKET** softkey.

- Enter the X- and Y-axes coordinates for the pocket's 1ST corner.

- Now enter either the size of the pocket or the coordinates for the 2ND corner.

    Size
    　　Enter the length of the pocket along the X- and Y-axes.

    2ND Corner
    　　Enter the X- and Y-axes coordinates for the 2ND corner. (The 2ND corner must be located diagonally from the 1ST corner.)

- Enter a depth (if needed).

- For "Direction," press either the **CW** softkey for a clockwise cutting direction or the **CCW** softkey for a counter-clockwise cutting direction.

- Enter the feed rate.

- If you want to program a corner blend radius, tilt angle and/or finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

  Corner Blend Radius
  You can add a corner blend radius to the corners of a rectangular pocket.

  Highlight the "Corner Blend Radius" field and enter a radius.
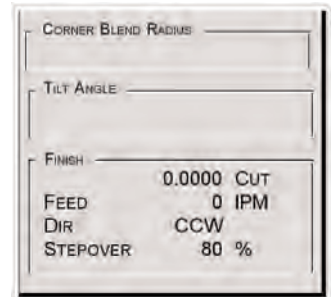
  Tilt Angle
  You can tilt a rectangular pocket by identifying a tilt angle.

  Highlight the "Tilt Angle" field and enter an angle (measured from the X-axis).

  Finish
  Finish allows you to leave some excess material that will be removed during the finish cut reducing, if not eliminating tool marks. This finish cut will automatically arc on and arc off.

  Enter the amount of material to be removed during the finish cut.

  Enter the feed rate for the finish cut.

  Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

The corner blend radius defaults to the last radius entered. Check the value to ensure that it is correct.

Enter a step over percentage (how much you want the tool to overlap on each pass).

Press the **USE** key.

Note:　If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Frame

When you program a rectangular frame, you define it by its first corner, and its size or diagonal corner.

Outside Frame

Inside Frame
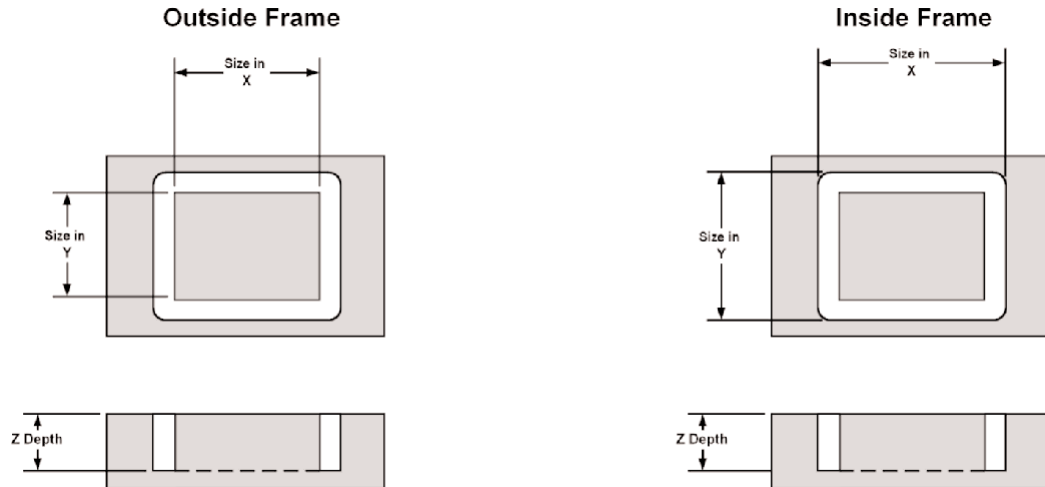
To program a rectangular frame:

- Press the **RECT** key.

- Press the **FRAME** softkey.

- Enter the X- and Y-axes coordinates for the frame's 1ST corner.

- In the "Size" field, enter the length of the frame along the X- and Y-axes (unless you are programming a 2ND corner).

- Enter the depth.

- Enter a corner blend radius (optional).

- For "Direction," press either the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

- Select a tool offset by pressing the appropriate softkey.

- Enter the feed rate.

- If you need to program a 2^(ND) corner and/or a tilt angle or would like to program a finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

2^(ND) Corner
Enter the X- and Y-axes coordinates for the 2^(ND) corner. (The 2^(ND) corner must be located diagonally across from the 1^(ST) corner.)

Tilt Angle
Enter the tilt angle.

Finish
Highlight "Cut" in the "Finish" field. Enter the amount of material to be removed during the finish cut.

Enter the feed rate for the finish cut.

For "Direction," press either the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Face

The "Rectangular Face" step provides a quick way to face off the workpiece. Simply enter the coordinates from one corner and either the size of the area to be faced off or the coordinates for a diagonal corner. **MILL*PWR*** will position the table at the lower left end of the area you have programmed.

To program a rectangular face:

- Press the **RECT** key.

- Press the **FACE** softkey.

- Enter the X- and Y-axes coordinates for the face's 1ST corner.

- In the "Size" field, enter the length of the face along the X- and Y-axes (unless you are programming a 2ND corner).

- Enter the depth.

- Select a tool offset by pressing the appropriate softkey.

- Enter the feed rate.

- If you wish to program a 2ND corner and/or a tilt angle or would like to program a finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

   2ND Corner
   Enter the X- and Y-axes coordinates for the 2ND corner. (The 2ND corner must be located diagonally across from the 1ST corner.)

   Tilt Angle
   Enter the tilt angle.

Finish

Enter a step over percentage (how much you want the tool to overlap on each pass).

• Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Slot

You can program a slot two ways:

- By entering the center point of each arc and the slot's width

- By entering the center point of one arc, the length and width of the slot, and the tilt angle

Choose a method based upon the information available from the print.
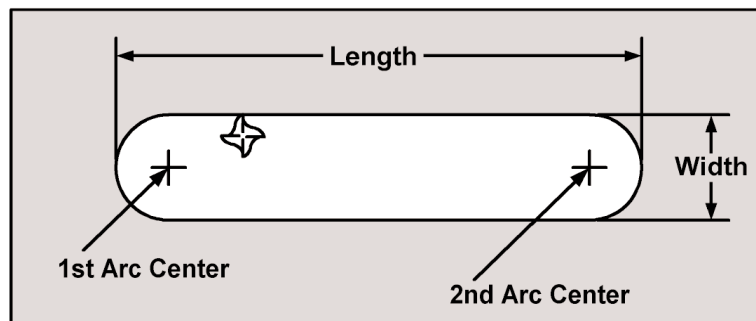
To program a slot:

- Press the **RECT** key.

- Press the **SLOT** softkey.

- Enter the X- and Y-axes coordinates for the "1ST Arc Center."

- Enter the X- and Y-axes coordinates for the "2ND Arc Center" or press the **MORE** softkey and enter the "Slot Length" and "Tilt Angle."

- Enter the depth.

- Select the direction you want the slot to be cut. Press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

- Enter the slot's width.

• Enter the feed rate.

• If you want to program a finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

Finish

Enter the amount of material to be removed during the finish cut.

Enter the feed rate for the finish cut.

Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

Enter a stepover percentage (how much you want the tool to overlap on each pass).

Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Circular Milling Functions

CIRCLE ⊕   MILL*PWR* offers several circular milling functions that let you program pockets, frames and rings quickly and easily.

POCKET	FRAME	RING			

Pocket

A pocket is a cavity or area on a part where material is removed when you machine. You can program a circular pocket by indicating the center point and radius.

To program a circular pocket:

- Press the **CIRCLE** key.

- Press the **POCKET** softkey.

- Enter the X- and Y-axes coordinates for the pocket's center point.

- Enter the depth.

- Enter the radius.

- For the "Direction," press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

.

• Enter the feed rate.

• If you want to program a finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

Finish

Enter the amount of material to be removed during the finish cut.
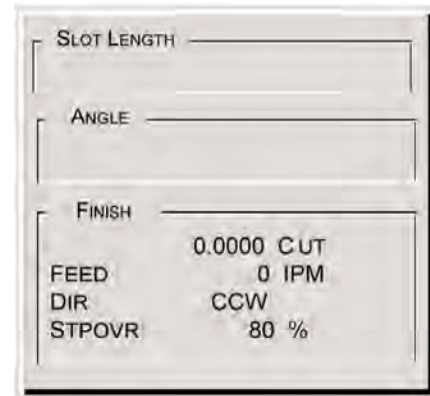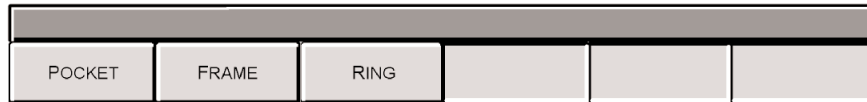
Enter the feed rate for the finish cut.

Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

Enter a step over percentage (how much you want the tool to overlap on each pass).

Press the **USE** key.

> FINISH
>
> 0.0000 CUT
> FEED   0 IPM
> DIR   CCW
> STPOVR   80 %

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Frame

A circular frame is determined by its center point and radius. The direction will determine if you are climb cutting or conventional cutting. The tool offset will determine if you are cutting an inside or outside frame.

INSIDE
FRAME

Radius

Z Depth

OUTSIDE
FRAME
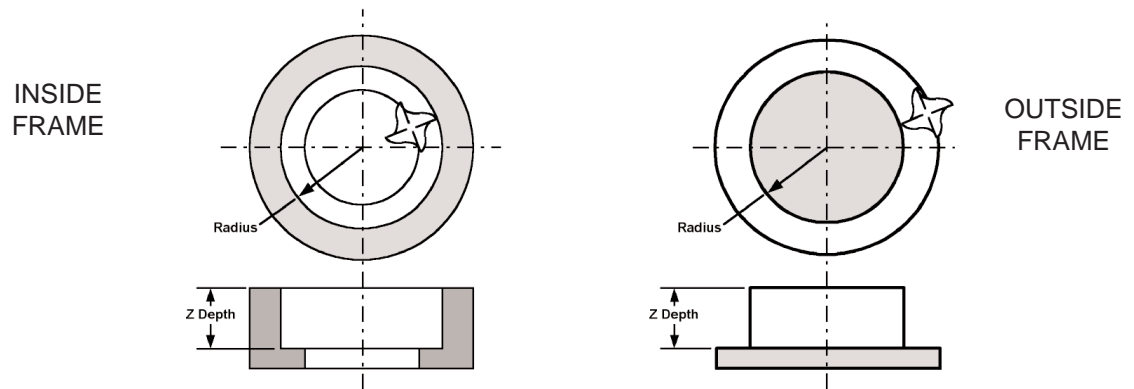
Radius

Z Depth

To program a circular frame:

• Press the **CIRCLE** key.

• Press the **FRAME** softkey.

• Enter the X- and Y-axes coordinates for the center point.

• Enter the depth.

• Enter the radius.

• For the direction, press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

- Select the appropriate tool offset.

- Enter the feed rate.

- If you would like to program a finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

  Finish
    Enter the amount of material to be removed during the finish cut.

    Enter the feed rate for the finish cut.

    Select the finish cut's direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.
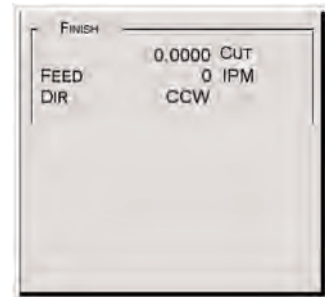
    Press the **USE** key.

Note: If the tool size and type in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Ring

A ring is a circular pocket with a circular
island in the center. A ring is determined by
its center point, outside radius (radius of the
pocket) and inside radius (radius of the
island).

The direction of the cut on the inside radius
will determine whether you are climb cutting
or conventional cutting. **MILL*PWR*** will reverse
the tool direction on the outside radius so that
the cutting direction stays the same.
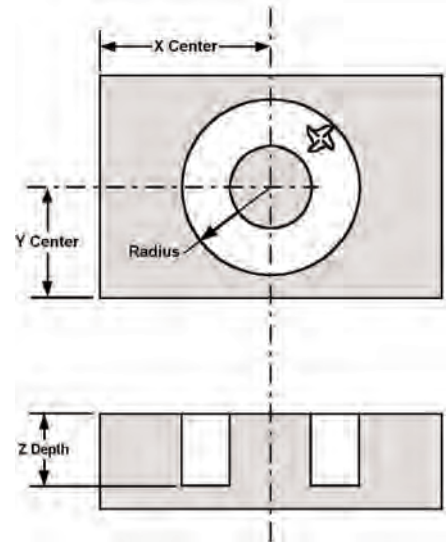
To program a ring:

- Press the **CIRCLE** key.

- Press the **RING** softkey.

- Enter the X- and Y-axes coordinates for the center
  point.

- Enter the depth.

- Enter the circle's outside and inside radii.

- For "Direction," press the **CW** softkey for
  clockwise or the **CCW** softkey for a counter-clock-
  wise direction.

- If you would like to program a finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

```
  FINISH
                    0.0000  CUT
  FEED                   0  IPM
  DIR                  CCW
  STPOVR               80  %
```

   Finish

   Enter the amount of material to be removed during the finish cut.

   Enter the feed rate for the finish cut.

   Select the finish cut's direction. Press the **CW** soft-key for clockwise or the **CCW** softkey for a counter-clockwise direction.

   Enter a step over percentage (how much you want the tool to overlap on each pass).

   Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Hole Patterns

HOLES   **MILL***PWR* includes several built-in routines that let you program hole patterns quickly and easily.

| ROW | FRAME | ARRAY | BOLT CIRCLE | | |
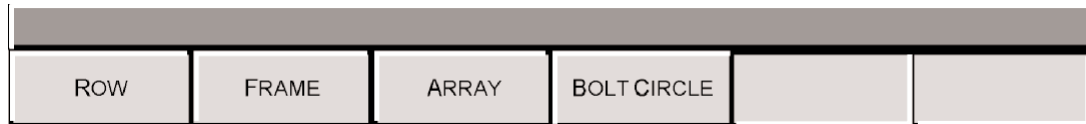
Row of Holes

A row of holes, can be programmed two ways:

*   By entering the coordinates of the first and last hole

*   By entering the coordinates of the first hole, the spacing between each hole and the row's tilt angle
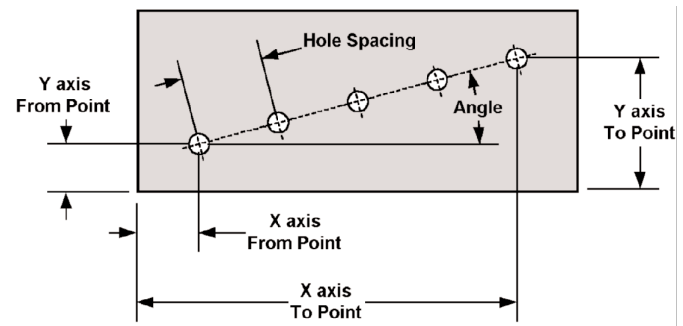
The "From" point refers to the center of the first hole, while the "To" point is the center of the last hole. Any additional holes will be spaced equally between these two.

To program a row of holes:

*   Press the **HOLES** key.

*   Press the **ROW** softkey.

*   In the "From" field, enter the X- and Y-axes coordinates for the center of the first hole.

*   Now either:

    Enter the X- and Y-axes coordinates for the center of the last hole in the "To" field; or

    Enter the distance you want between each hole (from center point to center point) in the "Hole Spacing" field. Also enter the tilt angle of the row of holes.

• Highlight the "Depth" field and enter the depth for each hole.

• Enter the number of holes you want to include in the row.

• Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Hole Frame and Hole Array

Hole frame and hole array patterns require the same information, but their patterns differ slightly. Hole frames (shown on the left) limit holes to the outside edge of a rectangular shape, while hole arrays (shown on the right) allow holes along the outside edge and throughout the center.
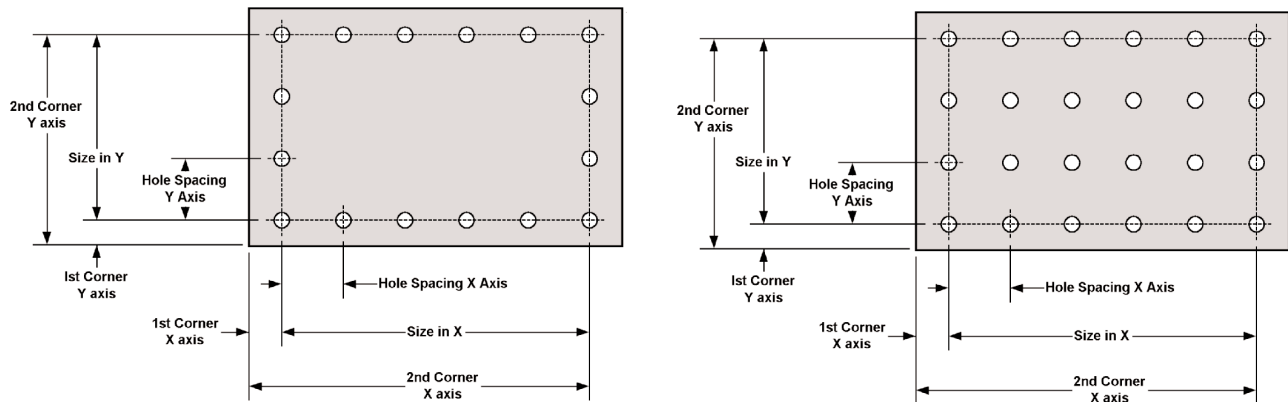
Hole frames and hole arrays can be defined three ways:

- By the position of the 1ST corner, size, and the number of holes

- By the position of the 1ST corner, position of the 2ND (diagonal) corner, and number of holes

- By the position of the 1ST corner, hole spacing, and number of holes

Choose the method that's easiest for you based upon the information from the print.

To create a hole frame or hole array:

- Press the **HOLES** key.

- Press the **FRAME** softkey to program a hole frame; press the **ARRAY** softkey to program a hole array.

- Enter the X- and Y-axes coordinates for the 1ST corner into the "From" field.

- In the "Size" field, enter the lengths along the X- and Y-axes.

- Enter the depth of the cut.

- In the "Holes" field, enter the number of holes you want to include along the X- and Y-axes.

- If you would like to program a 2ND corner, hole spacing and/or tilt angle, press the **MORE** softkey. Otherwise, press the **USE** key.

  2ND Corner
    Enter the X- and Y-axes coordinates for the 2ND corner (located diagonally from the 1ST corner).

  Hole Spacing
    Specify the hole spacing between holes along the X- and Y-axes.

  Tilt Angle
    Enter a tilt angle, if needed.

  Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Bolthole Circle Patterns

A bolthole circle pattern is defined by its center point, radius and number of holes. You can program partial bolthole patterns by pressing the **MORE** softkey and entering a start angle and an end angle.

To program a bolthole circle pattern:

- Press the **HOLES** key.

- Press the **BOLT CIRCLE** softkey.

- Enter the X- and Y-axes coordinates for the bolthole circle pattern's center point.

- Enter the depth of cut for the holes.

- Enter the radius of the hole pattern.

- For "Direction," select the direction you want to cut. Press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

- Enter the number of holes you want to include in the circle.
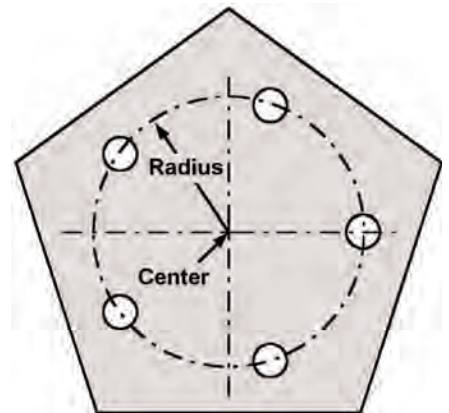
• If you need to program a partial bolthole circle pattern, press the **MORE** softkey. Otherwise, press the **USE** key.

START ANGLE

END ANGLE

Enter the start angle and end angle of the first and last holes in the bolthole circle pattern.

Press the **USE** key.

90.0°
Start Angle
End Angle
Left Blank

90.0°
Start Angle
-90.0°
End Angle

90.0°
Start Angle
End Angle
Set to 0°
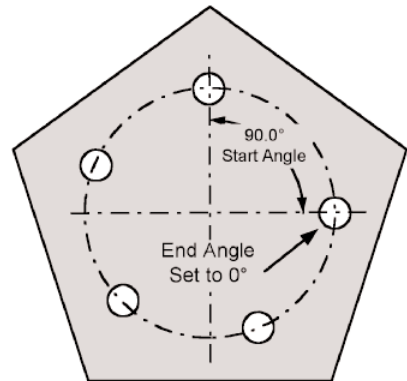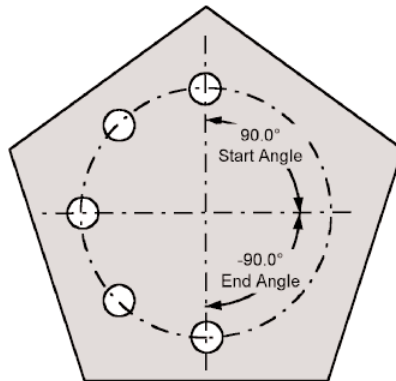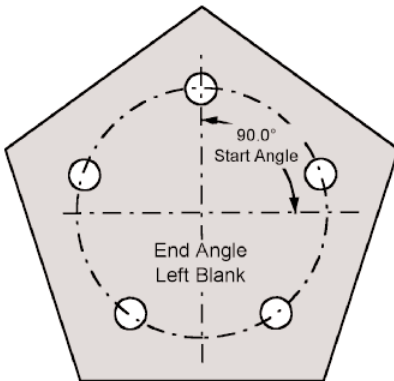
Note: If the tool size and type in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Additional Milling Functions

Additional milling functions are available from the PGM screen by pressing the **MORE STEPS** softkey.
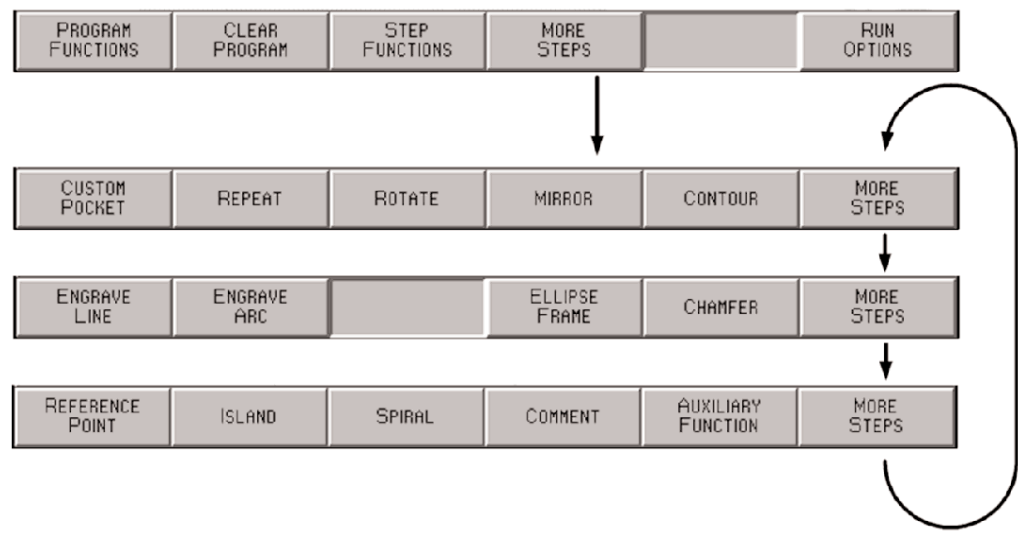
Custom Pocket

Note: The custom pocket step must immediately follow a closed contour.

You can create a custom pocket from any closed contour. A closed contour is any shape consisting of lines, arcs, and/or blends, where the last step ends at the same point where the first step begins. **MILL*PWR*** will indicate a closed contour with double lines to the right of the applicable steps in the program list.

The "Custom Pocket" step must be placed immediately following the last step of the closed contour. **MILL*PWR*** will automatically fill in the step range for you. You'll still need to fill in the "Entry Point," which is the plunge point for the tool, and set the feed rate for the custom pocket. Custom pockets can be cut from the center (inside) of the pocket out or from th edge (outside) of the pocket in (towards the center). The finish cut will occur after the material from the inside of the pocket has been removed.

The direction of the cut is a direct result of the order of the steps entered to shape the contour.

To program a custom pocket:

- Create a closed tool path.

- Position the cursor immediately below the closed tool path.

- Press the **MORE STEPS** softkey.

- Press the **CUSTOM POCKET** softkey. **MILL*PWR*** will automatically fill in the step range for you.

- Enter the X- and Y-axes coordinates for the entry (plunge) point.

- Enter the feed rate.

- Press either the **OUTSIDE** or **INSIDE** softkey. "Outside" clears the custom pocket from the outside in. "Inside" clears the custom pocket from the inside out.

- Press **USE** or continue on to finish cut.

- If you would like to program a finish cut, enter the amount of material to be removed during the finish cut in the "Cut" field.

    Finish
        Enter the feed rate for the finish cut.

        For "Direction," press either the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

        Enter the step over percentage (how much you want the tool to overlap on each pass).

        Press the **USE** key.

Repeat

Using this step you can repeat whole programs or sections of programs horizontally, vertically or both.
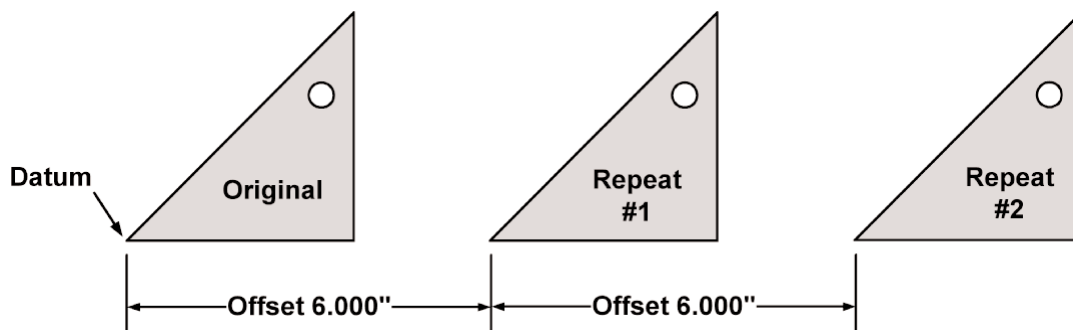
To program a repeat:

- From the PGM screen, press the **MORE STEPS** softkey.

- Press the **REPEAT** softkey.

- Enter the number of the first step and the last step in the step range that you want to repeat.

Tip: Use the **TEACH POSITION** softkey to indicate the X and Y offsets.

- Enter the offset along the X- and/or Y-axes. (The *offset* is the distance between repeats.)

- Enter how many times you want to repeat the original shape.

Note: The steps being repeated must precede the "Repeat" step.

Note: Entering a 0 (zero) for the number of repeats will change the location of the original steps entered in the FROM and TO fields. To shift the datum of an entire program, place a repeat step at the end of the program and enter the required coordinate shift(s) in the X-, Y-, and/or Z-axis fields. When USE is pressed the entire program will shift by the entered values.

Rotate

With "Rotate," you can rotate whole programs or sections of programs.


```
001    TOOL
002 ⊓  MILL LINE
003 ‖  MILL LINE
004 ⊔  MILL LINE
005    POSITION / DRILL
006    ROTATE
```

Note: The steps rotated must precede the "Rotate" step.
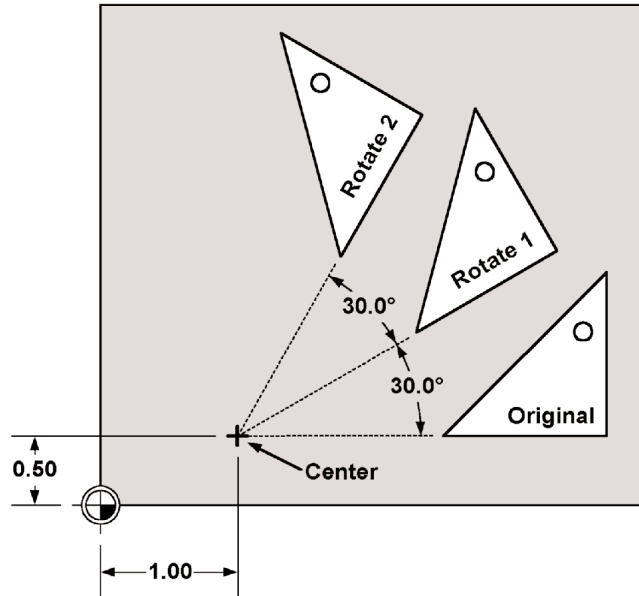
To program a rotate:

- From the PGM screen, press the **MORE STEPS** softkey

- Press the **ROTATE** softkey.

- Enter the first and last steps in the range of steps that you would like to rotate.

- Enter the X- and Y-axes coordinates for the center point of rotation.

- Enter the angle (in degrees) for each rotation.

- Enter the number of times you want to rotate the shape. If you want to rotate the program or sections of the program you are working with, enter "0." If you want additional rotations of the original, enter the number of rotations that you want in addition to the original.

- Press the **USE** key.

Mirror

With "Mirror," you can create a mirror image of an entire program or a section of a program.

Note:　The steps being mirrored must precede the "Mirror" step.

To program a mirror:

- From the PGM screen, press the **MORE STEPS** softkey.

- Press the **MIRROR** softkey.

- Enter the number of the first step and the last step in the step range that you want to mirror.

- Define the axis of reflection (a line that separates the mirrored image from the original one).

  For "1ST Axis Point," enter the X- and Y-axes coordinates for the axis of reflection.

  For "2ND Axis Point," enter the X- and Y-axes coordinates for the axis of reflection.

- Press the **USE** key.

Contour

The "Contour" step enables you to approach and/or depart from the part on a straight line or with an arc.

Conditions: 1) The contour step must immediately follow the "Contour" steps.

   2) Contours can only be associated with Lines and Arcs.

By adding contours before and/or after a continuous tool path, you will avoid starts and stops striking against the workpiece edge.

With an arc approach/departure, the tool will take a rounded turn as it nears or exits the workpiece. (The curved dotted line on the left of the graphic represents an arcing approach to the workpiece.)

With a straight approach/departure, the tool path is extended away from the workpiece. (The dotted line on the right of the graphic shows a straight contour at the end of the cutting path.)
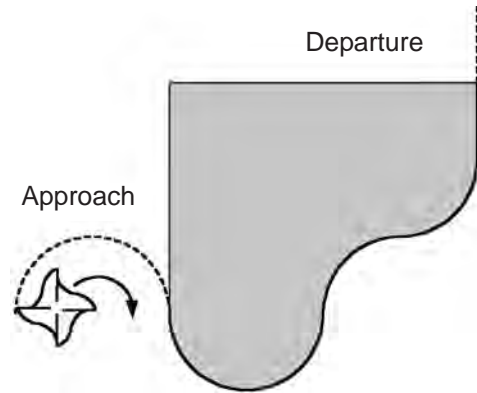
When you program a contour, you will be asked for the step range of the continuous tool path, plus the approach and/or departure types and distances. You will also have the option of programming a finish cut.

The step range can include one or multiple steps. If you are planning to add a contour to an individual step, the first and last steps in the range should be the same.

Because the approach and departure fields are independent of each other, you may select one or both for the step range you have chosen. Select "None" as the type for whichever option you don't want.

To program a contour:

• From the PGM screen, highlight the step below the *last* step in the continuous contour.

• Press the **MORE STEPS** softkey.

• Press the **CONTOUR** softkey.

• Notice that "First" and "Last" in the "Step Range" field will be filled in for you.

• If you wish to program an approach, press the **STRAIGHT** or **ARC** softkey as an approach type. Otherwise, press the **NONE** softkey.

   Enter how far from the part you want the approach to begin.

• To program a departure, press the **STRAIGHT** or **ARC** softkey as the departure type. Otherwise, press the **NONE** softkey.

   Enter how far from the part you want the tool to travel.

• If you would like to program a finish cut, enter the amount of material to be removed during the finish cut. Otherwise, press the **USE** key.

   Finish
      Enter the feed rate.

      Using the softkeys, select the tool direction (**FORWARD** or **REVERSE**).

      Press the **USE** key.

   Note: If a finish cut is programmed with a reverse cut direction, any departure will be ignored. The tool will automatically return to it's previous position.

Engrave

With **MILL*PWR***, you have the ability to engrave letters, numbers and symbols, along a straight line or on an arc. Choose from a simple, stick or stencil font. The character height, font and modifier settings you select will define the engraving's appearance.

Condition: The tool diameter being used establishes the spacing between letters.
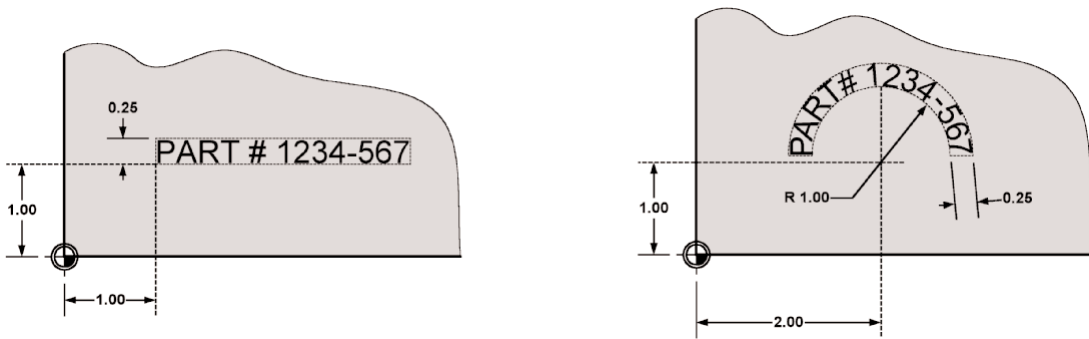
Engrave Line

To engrave along a line:

- From the PGM screen, press the **MORE STEPS** softkey.

- Press the **MORE STEPS** softkey again.

- Press the **ENGRAVE LINE** softkey.

- Enter the X- and Y-axes coordinates for the point at the lower left corner of the engraving.

- Enter the character height.

- Enter the tilt angle (if any).

- Enter the depth.

> *Use "mirrored" if you are making a mold.*

- Highlight "Font". If you want to change the font, press the **ENGRAVER FONTS** softkey, highlight the font you want, then press **ENTER**.

- Highlight "Modifier" and press either the **NORMAL** or **MIRRORED** softkey. "Normal" means that the engraving will be readable from left to right; "Mirrored" will make the engraving appear backwards.

- Enter the feed rate.

- Press the **EDIT TEXT** softkey.

- Notice that the screen is divided into two sections— "Text Block" and "Characters." You can switch from one to the other by pressing either the **TEXT BLOCK** or **CHARACTER** softkey. Use the arrow keys to move around within each section.

OFF	FEED: 0.0 100%		SCALE: 1.000	0:00	INCH

TEXT BLOCK

PART # 1234-567

CHARACTERS

```
 A  B  C        A  B  C
 D  E  F  G  H   D  E  F  G  H      7  8  9  /     (  )  $  #  @  ^
 I  J  K  L  M   I  J  K  L  M      4  5  6  ×     [  ]  %  &  |  \
 N  O  P  Q  R   N  O  P  Q  R      1  2  3  -     <  >  ,  :  ;  ?
 S  T  U  V  W   S  T  U  V  W      0  .  =  +     {  }  !  '  "  ~
 X  Y  Z         X  Y  Z
```

TEXT BLOCK	CHARACTERS	DELETE LINE	INSERT LINE	CLEAR ALL	EDIT TEXT

- In the "Character" section, highlight a letter, number, symbol or space. Press the **ENTER** key to make a selection. (The alphabet block on the left side represents capital letters; the block on the right side represents lowercase letters.)

  To add a space, move the cursor to a blank spot anywhere within the "Characters" box and press the **ENTER** key.

  You can edit text by pressing the **TEXT BLOCK** softkey. Highlight the character you want to edit, and then press one of the following:

  The **CLEAR** key will delete a character.

  The **DELETE LINE** softkey will erase the entire line of text that the cursor is in and move all following text lines up.

  The **INSERT LINE** softkey will insert a blank text line, moving existing text lines down.

  The **CLEAR ALL** softkey will erase all of the text within the text block.

- When you have finished creating/editing the text, press the **EDIT TEXT** softkey (or press the **USE** key) to return to the "Engrave" screen. **MILL*PWR*** will display the text that you have created on the right.

- Press the **EDIT TEXT** softkey if you want to make changes, or press the **USE** key to add the engraving step to the program.

- Press the **EDIT TEXT** softkey when you have finished entering the characters.

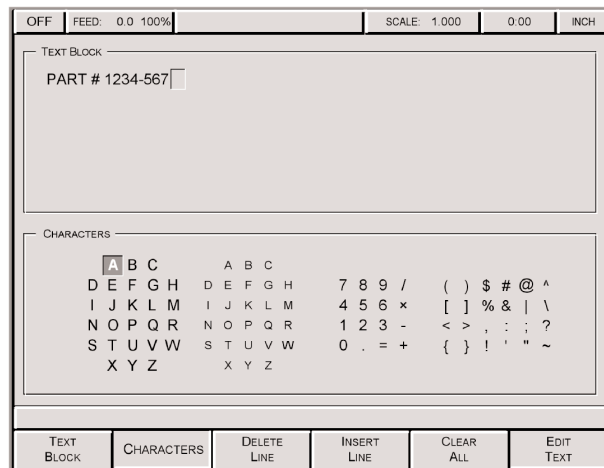- Press the **USE** key.

Engrave Arc

To engrave along an arc:

- From the PGM screen, press the **MORE STEPS** softkey.

- Press the **MORE STEPS** softkey again.

- Press the **ENGRAVE ARC** softkey.

- Enter the X- and Y-axes coordinates for the center point of the engraving's arc.

- Enter the character height.

- Enter the radius.

- Select either the **UP** or **DOWN** softkey. "Up" means that the arc will curve upward; "Down" means that the arc will curve downward.

- Enter the tilt angle (if any).

- Enter the depth.

- Highlight "Font." Press the **ENGRAVER FONTS** softkey, highlight the font you want, then press **ENTER**.

- Highlight "Modifier." Press either the **NORMAL** or **MIRRORED** softkey. "Normal" means the type will be readable from left to right; "Mirrored" will make the engraving appear backwards.

- Enter the feed rate.

- Press the **EDIT TEXT** softkey.

Downward Arc

Upward Arc
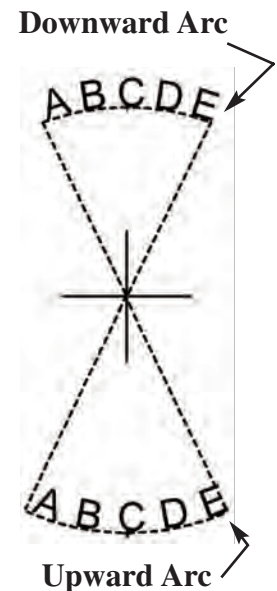
ENGRAVE ARC	006

TEXT CENTER
X   2.0000 ABS
Y   1.0000 ABS

CHARACTER HEIGHT
0.2500

RADIUS
1.0000 Up

TILT ANGLE
0°00'00"

DEPTH
0.0200 ABS

STYLE
SIMPLE FONT
NORMAL MODIFIER

TOOL
FEED   IPM

Use "mirrored" if you're making a mold.

- Notice that the screen is divided into two sections— "Text Block" and "Characters." You can switch from one to the other by pressing either the **TEXT BLOCK** or **CHARACTER** softkey. Use the arrow keys to move around within each section.

- In the "Character" section, highlight a letter, number, symbol or space. Press the **ENTER** key to make a selection. (The alphabet block on the left side represents capital letters; the block on the right side represents lowercase letters.)

    To add a space, move the cursor to a blank spot anywhere within the "Characters" box and press the **ENTER** key.

    You can edit text by pressing the **TEXT BLOCK** softkey. Highlight the character you want to edit, and then press one of the following:

    The **CLEAR** key will delete a character.

    The **DELETE LINE** softkey will erase the entire line of text that the cursor is in and move all following text lines up.

    The **INSERT LINE** softkey will insert a blank text line, moving existing text lines down.

    The **CLEAR ALL** softkey will erase all of the text in the text block.

- When you have finished creating/editing the text, press the **EDIT TEXT** softkey (or press the **USE** key) to return to the "Engrave" screen. **MILL*PWR*** will display the text that you have created on the right.

- Press the **EDIT TEXT** softkey if you want to make changes, or press the **USE** key to add the engraving step to the program.

- Highlight a letter, number or symbol. Press the **ENTER** key after each selection. (The alphabet block on the left represents capital letters; the block on the right represents lowercase letters.)

To add a space, move the cursor to a blank spot anywhere within the "Characters" box and press the **ENTER** key.

- Press the **EDIT TEXT** softkey when you have finished entering the characters.

- Press the **USE** key.

Ellipse Frame

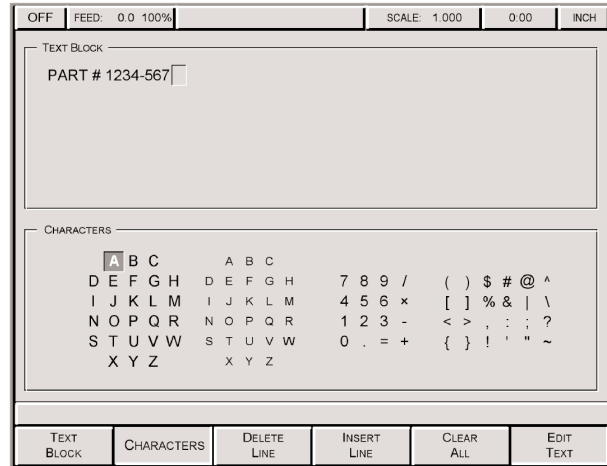An ellipse frame is a closed curve with an oval shape. The only offset available for this is center offset. Due to the nature of an ellipse, when the center of the tool tracks an ellipse, its edge will have some error. To minimize the amount of error and cut the most accurate possible ellipse, use the smallest tool diameter available.

To cut an inside ellipse, subtract the tool diameter from the size in both X-and Y-axes.

To cut and outside ellipse, add the tool diameter to the size in both X-and Y-axes.
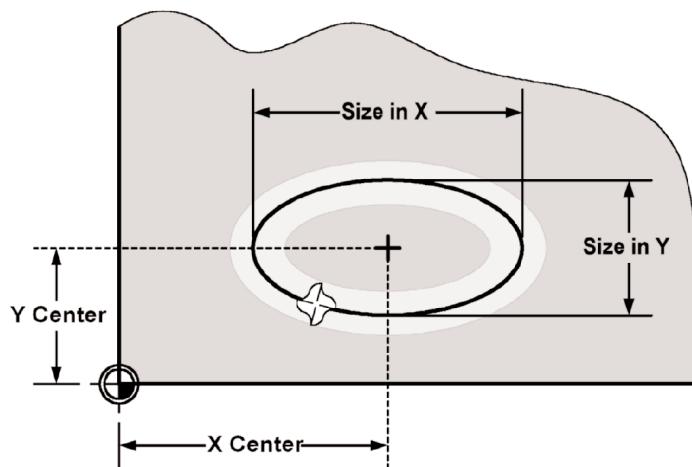
To program an ellipsed frame:

- From the PGM screen, press the **MORE STEPS** softkey.

- Press the **MORE STEPS** softkey again.

- Press the **ELLIPSE FRAME** softkey.

- Enter the location of the center point for the X-and Y-axes.

- Enter the size along the X-and Y-axes.

- Enter the depth.

- Select the cutting path direction. Press the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

- Enter the feed rate.

- If you need to program a tilt angle press the **MORE** softkey.

    Tilt Angle
        Highlight "Tilt Angle" and enter the angle (from the X-axis).

- Press the USE key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool
        settings before running the program (refer to **Programming a Tool Step**).

Chamfer

A chamfer is a bevel or line that's inserted between two lines to relieve sharp angles or corners on a part.

You can insert a chamfer between two intersecting lines whose steps are adjacent in the program step.

There are three ways you can program a chamfer:

- With two lengths

- With Length 1 and an angle

- With Length 2 and an angle
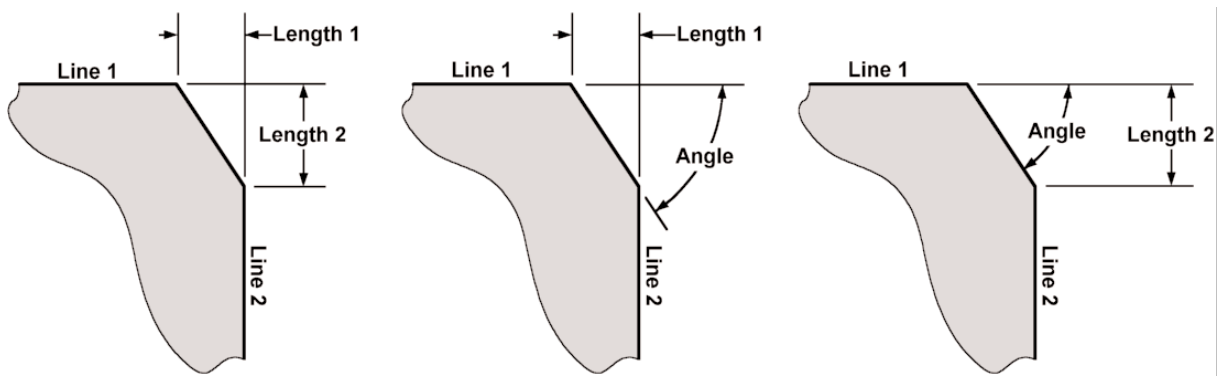
Choose a method based upon information from the print. As you are programming, keep in mind that the angle is measured from the line along the X-axis and sweeps towards the adjacent line. You may also close a contour with a chamfer (e.g., a triangle) by inserting the chamfer step immediately after the last step in the contour.

To program a chamfer:

- From the PGM screen, locate the lines you want to insert a chamfer between. Highlight the second line.

- Press the **MORE STEPS** softkey.

- Press the **MORE STEPS** softkey again.

- Press the **CHAMFER** softkey.

- **MILL*PWR*** will automatically fill in the "From" and "To" fields in "Steps" for you. (Press the **CLOSE CONTOUR** softkey if you want to chamfer the end of a contour with the beginning.)

- Enter the distance from the common point of both lines. ("Length 1" refers to the line listed in the "From" field; "Length 2" refers to the line listed in the "To" field.) If you plan to use an angle, enter only one length.

- If you entered one length above, highlight the "Angle" field and enter the chamfer's angle from the X-axis; otherwise, leave it blank.

- Adjust the feed rate, if necessary.

- Press the **USE** key.

CHAMFER	005
STEPS	
FROM	2
TO	4
SIZE	
	LENGTH 1
	LENGTH 2
ANGLE	
FEED RATE	
	10 IPM

Reference Point

Depicted as a plus sign (+) on the PGM screen, a reference point is a graphical representation of a coordinate in the program.

Reference points are commonly used to identify center points, tangent points and other part features. They can even be used as the basis for incremental moves.

As you program, note that placing a reference point in a continuous tool path will break the path. Otherwise, reference points do not affect your program's performance in any way—in fact, **MILL*PWR*** will skip over them altogether when you run a program.

There's no limit to the number of reference points you may use, so include as many as you need.

To program a reference point:

- From the PGM screen, press the **MORE STEPS** softkey.

- Press the **MORE STEPS** softkey two more times.

- Press the **REFERENCE POINT** softkey.

- Enter your reference point's position for X, Y and Z.

- Press the **USE** key.

Island

An island is a raised area within a custom pocket that remains after material has been removed from around all of its sides.

Though islands are easy to program, they must be placed correctly within the program sequence. Steps for the island's closed continuous tool path must appear first, followed by the island step. Steps for a custom pocket's closed continuous tool path must appear next, followed by the custom pocket step.

You may program more than one island within the custom pocket.
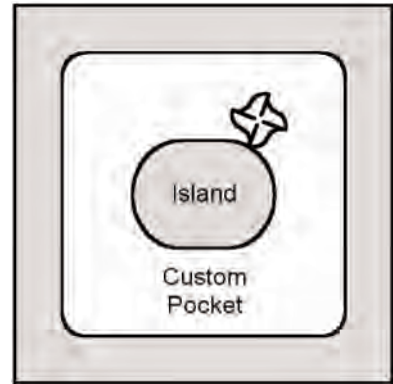
<u>To program an island:</u>

- Program a closed continuous tool path for the island.

- From the PGM screen, highlight the step after the <u>last</u> step of the island's closed continuous tool path.

- Press the **MORE STEPS** softkey.

- Press the **MORE STEPS** softkey two more times.

- Press the **ISLAND** softkey.

- Check that the first and last steps listed in the step range match the first and last steps for the island's closed continuous tool path.
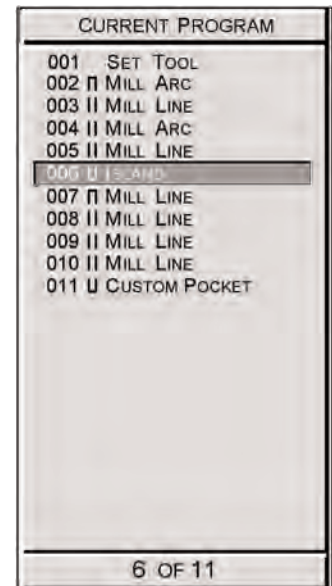
   If they're correct, press the **USE** key; if they're not, press the **CANCEL** key and check the closed continuous tool path for the island.

- Now you are ready to program a custom pocket.

To program a custom pocket:

- Program a closed continuous tool path for a custom pocket.

- Highlight the step after the last step of the custom pocket's closed continuous tool path.

- Press the **MORE STEPS** softkey.

- Press the **CUSTOM POCKET** softkey. **MILL*PWR*** will automatically fill in the step range for you.

- Enter the X- and Y-axes coordinates for the entry (plunge) point.

- Enter the feed rate.

- Press either the **OUTSIDE** or **INSIDE** softkey. "Outside" clears the custom pocket from the outside in. "Inside" clears the custom pocket from the inside out.

- If you would like to program a finish cut, enter the amount of material to be removed during the finish cut in the "Cut" field. Otherwise, press the **USE** key.

  Finish
  Enter the feed rate for the finish cut.

  For "Direction," press either the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

  Enter the stepover percentage (how much you want the tool to overlap on each pass).

  Press the **USE** key.

Spiral

A spiral is a winding and gradually widening curve or coil. Spirals are defined by their center point, beginning and ending radii, and sweep angle.

The center point is the (X, Y) coordinate at the core of the spiral. The beginning radius is the distance from the spiral's center point to its starting point, and the ending radius is the distance from the spiral's center point to its ending point.
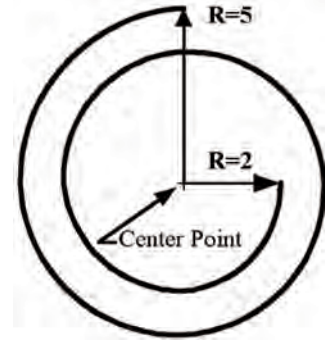
To program a spiral:

- From the PGM screen, press the **MORE STEPS** softkey.

- Press the **MORE STEPS** softkey two more times.

- Press the **SPIRAL** softkey.

- Enter the spiral's center point.

- Enter the depth.

- Enter a start and end radius (measured from the center point). If the ending radius is less than the start radius, then the tool will cut from the outside in, or vice versa.

- Select the direction by pressing the **CW** softkey for a clockwise direction or the **CCW** softkey for a counter-clockwise direction.

As you are programming, keep in mind that you may "teach" the positions for the center point and depth.

- Enter the sweep angle (e.g., 180 for a half-rotation; 360 for one rotation; 1080 for three rotations, etc.).

- Using the softkeys, choose a tool offset—**LEFT**, **CENTER**, **RIGHT**, **INSIDE** or **OUTSIDE**.

- Enter a feed rate.

Use the calculator keys to let MILLPWR factor the angles for you. For example, enter "3.5 x 360" to create a spiral with three and a half rotations.

• If you need to program a start angle or would like to program a finish cut, press the **MORE** softkey. Otherwise, press the **USE** key.

Start Angle
You will want to program a start angle if the beginning point of your spiral on the print is tilted—that is, not positioned at a 0 degree angle (3 o'clock).

Highlight "Start Angle," then enter the angle of the spiral's starting position (measured from the X-axis).
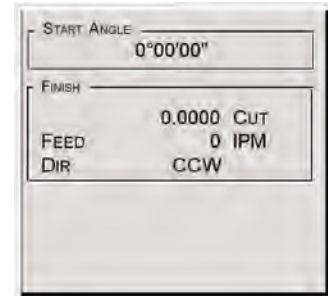
Finish
Enter the amount of material to be removed during the finish cut.

Enter the feed rate for the finish cut.

Select the direction for the finish cut. Press the **CW** softkey for clockwise or the **CCW** softkey for a counter-clockwise direction.

Press the **USE** key.

Note: If the tool size and type listed in the "Tool" field are incorrect, change the tool settings before running the program (refer to **Programming a Tool Step**).

Comment Step

With **MILL***PWR*, you have the ability to insert messages
anywhere within a program. These messages can be
displayed during machining (at run-time) or as Operator
Intervention Messages (OIM). These messages become
operational steps within the program and communicate
pertinent information—like "ROTATE PART" or
"ACTIVATE COOLANT".

For comments that don't require an operator intervention,
select "No" when asked if you want the comment displayed at run-time, and **MILL***PWR*
will skip over them during machining. (You can always retrieve the message by
highlighting the comment step in the program steps list and pressing **ENTER**.)

To program a comment step:

- From the PGM screen, press the **MORE STEPS**
  softkey.

- Press the **MORE STEPS** softkey two more times.

- Press the **COMMENT** softkey.

- Enter a message. You may include up to 20
  characters, mixing numbers, letters, spaces and
  symbols if you wish.

  To add a space, highlight a blank spot within the
  alphabet box and press the **ENTER** key.

  To add numbers, math symbols (+, -, x, ÷) or a
  decimal point, use the keypad on the operator console.

- Press the **ALPHABET** softkey again when you have finished entering a message, and
  the alphabet box will disappear.

- Highlight "Display At Run-time". Press the **YES** softkey to display the message
  during machining or the **NO** softkey if you don't want the message displayed.

- Press the **USE** key.
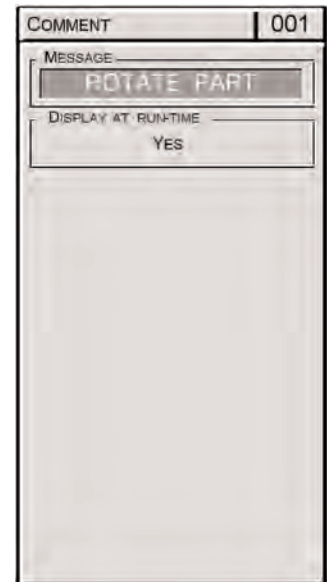
Auxiliary Function (AMI Option)

If you purchased an optional AMI (Auxiliary Machine Interface) from ACU-RITE, you can program coolant pumps, automatic lubrication systems and other devices to turn on, off or pulse automatically. Simply insert an "Auxiliary Function" step into your program each time you want to change a device's status.

Note: Before you can use the Auxiliary Function feature, the AMI and the output devices you're controlling must be properly installed (refer to the **AMI Reference Manual** for instructions).

Pausing your program won't affect the auxiliary functions. Each device will continue to run normally. However, **MILL*PWR*** will shut off all of the relays automatically when the program ends. To shut off a device earlier in the program, insert another auxiliary step with the appropriate relay turned "off."

Devices programmed to switch "on" will run continuously as you machine; devices set to "pulse" will run for the length of time (up to 999.9 seconds) you define.

To add an auxiliary function step to your program:

• Check that the AMI is properly connected to **MILL*PWR*** and that each device is connected to the AMI's J18 output connector (refer to the AMI Reference Manual).

• From the PGM screen, highlight a step within the program where you want to add or insert an auxiliary function step.

• Press the **MORE STEPS** softkey.

• Press the **MORE STEPS** softkey two more times.

- Press the **AUXILIARY FUNCTION** softkey.

- Highlight the auxiliary relay(s) that you want to program. (For example, "AUX 1" refers to the output device that's connected to pins 1 and 2 on the J18 connection.) Select from the **OFF**, **ON** and **PULSED** softkeys for the relay(s).

    ON: When the Auxiliary Function Step is executed, an output signal is generated. The signal will continue until another Auxiliary Function Step programmed to turn the signal OFF is executed , the **TABLE STOP** button is pressed, an E-Stop input signal is detected, the program is stopped, or the program ends.

    OFF: When the Auxiliary Function Step is executed, any current signal is turned off.

    Pulsed: When the Auxiliary Function Step is executed, an output signal is generated for the time duration specified. The signal will continue until the time duration ends, another Auxiliary Function Step programmed to turn the signal OFF is executed, the **TABLE STOP** button is pressed, an E-Stop input signal is detected, the program is stopped, or the program ends.

- If you selected "Pulsed," then enter the amount of time (in seconds) that you want the relay to remain on. The maximum amount of time that a relay can pulse is 999.9 seconds.

- Highlight the Display User Prompt field.

    NO: If you do not want an operator prompt to be displayed press **NO**.

    Before Step: Pressing Before Step causes an operator prompt to display before the auxiliary function(s) execute. The prompt will show you what each output will do. Pressing **GO** will execute the auxiliary function(s).

    After Step: Pressing After Step causes the operator prompt to display after the auxiliary function(s) execute. The program will pause until **GO** is pressed or a signal from an auxiliary device is received.

- Pressing "Yes" for the Set Continuous field allows an Auxiliary Function Step to be entered (e.g. activate coolant) without interrupting the continuous milling path.

- Press the **USE** key.

    Note: Check that the devices you are using are powered on before running the program.

Step Functions Softkey

Additional functions are available from the PGM screen by pressing the **STEP FUNCTIONS** softkey.

PROGRAM FUNCTIONS	CLEAR PROGRAM	STEP FUNCTIONS	MORE STEPS		RUN OPTIONS

EXPLODE	REVERSE STEP	REVERSE PATH	CHANGE STEPS	DELETE STEPS	COPY/MOVE STEPS
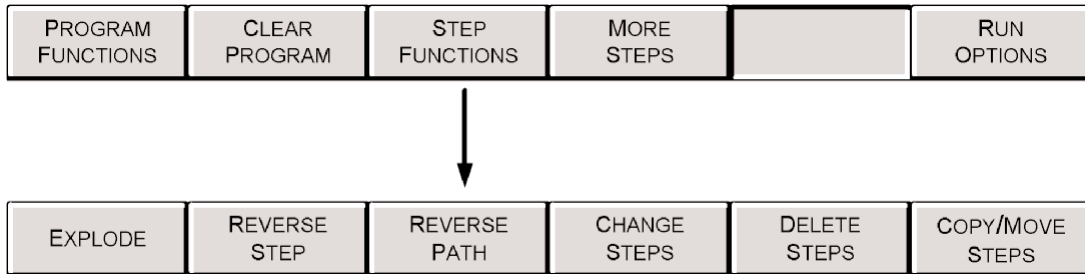
Explode

This feature will "explode" a program step into several, more detailed steps. You can explode the following functions:

- All "Holes" functions (row, frame, array and bolt circle)

- Repeat, Mirror and Rotate

- Engrave

In the example on the right, is a programmed bolt-hole circle with eight holes. Use the same basic pattern, but with seven holes.

To edit the program, first highlight the "Bolt Circle" step and then press the **EXPLODE** softkey.

CURRENT PROGRAM
001   SET TOOL
002   BOLT CIRCLE

2 OF 2
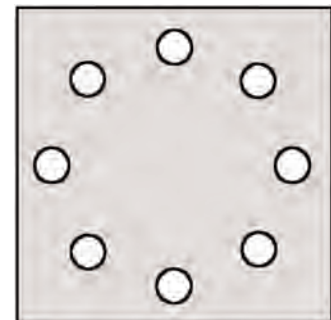
MILL*PWR* will explode the "Bolt Circle" steps into eight steps (shown on the left side, below). Next highlight the step for the hole that needs to be delete (shown in the center) and press the **CLEAR** key. As you can see in the graphic on the right, the bolt-hole circle pattern now has seven holes.

Notice how the individual Pos/Drill" steps replaced the Bolt Circle"step.

CURRENT PROGRAM	
001	SET TOOL
002	POSITION / DRILL
003	POSITION / DRILL
004	POSITION / DRILL
005	POSITION / DRILL
006	POSITION / DRILL
007	POSITION / DRILL
008	POSITION / DRILL
009	POSITION / DRILL

2 OF 2

CURRENT PROGRAM	
001	SET TOOL
002	POSITION / DRILL
003	POSITION / DRILL
004	POSITION / DRILL
005	POSITION / DRILL
006	POSITION / DRILL
007	POSITION / DRILL
008	POSITION / DRILL
009	POSITION / DRILL

4 OF 9
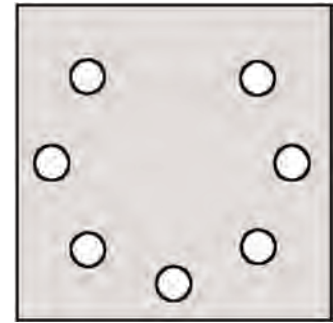
To explode a step:

- Highlight the step.

- Press the **STEP FUNCTIONS** softkey.

- Press the **EXPLODE** softkey.

The step that you exploded will be replaced with the individual lines, arcs or positions that made up the step.

Reverse Step

The reverse step option instantly switches the "From" and "To" points and tool offset.

To reverse a milling function:

- From the PGM screen, use the arrow keys to highlight the step that you want to reverse.

- Press the **STEP FUNCTIONS** softkey.

- Press the **REVERSE STEP** softkey.

Reverse Path

With the reverse path option, you can reverse any continuous tool path. This will especially come in handy when you are working with DXF files. As you import these types of files, **MILL*PWR*** will sort and then group the steps into a logical order, creating continuous paths. In some cases, the paths may need to be reversed after being imported so that the tool's offset, direction and beginning and end points satisfy the machining requirements.

As you become more familiar with reverse path, you will find other creative ways to use this feature. For instance, you can save time when cutting a part by using a heavy cutting tool and a conventional cut for a rough first pass. On the second pass, switch to a finish cutter, then copy and reverse the path for a climb cut on the finish pass.

To reverse a continuous tool path:

- From the PGM screen, highlight any step within the continuous tool path that you want to reverse.

- Press the **STEP FUNCTIONS** softkey.

- Press the **REVERSE PATH** softkey.

Change Steps

The "Change Steps" feature gives you the ability to change or edit the depth, offset and feed rate of several steps simultaneously.

Note: You can use this feature from anywhere within the program—you don't need to highlight a specific step within the step range.

To use the "Change Steps" feature:

• From the PGM screen, press the **STEP FUNCTIONS** softkey.

• Press the **CHANGE STEPS** softkey.

• Enter the first and last step numbers that you would like to change in the "Step Range" field.

• Highlight each field that you want to change and enter the new data.

• Press the **USE** key.

Note: It's a good idea to highlight the changed steps in the program sequence, then press the **ENTER** key. Each step should include the new settings.

Delete Steps

MILL*PWR* gives you the option of deleting steps in two ways: using the **DELETE STEPS** softkey or using the **CANCEL** key.

When deleting single steps, we suggest highlighting the step then pressing the **CANCEL** key.

When deleting a range of steps—after you have merged programs, for example—the "Delete Steps" feature is usually the best option.

To delete a group of steps from your program:

• From the PGM screen, press the **STEP FUNCTIONS** softkey.

• Press the **DELETE STEPS** softkey.

• In the "Step Range" field, enter the first and last step numbers that you would like to delete.

• Press the **USE** key.

Copy/Move Steps

Copy/Move operations make it easy for you to duplicate or rearrange steps within the program. You will find the "Move" feature especially useful for editing steps generated from a DXF file.

After you press the **COPY/MOVE STEPS** softkey, you will be asked to enter a step range, then either copy or move the steps.

Press the **MOVE** softkey to relocate the steps. Press the **COPY** softkey to create an identical copy of the steps you have chosen (the original steps will remain in place). After you press the **USE** key, the new steps will be inserted into the program.

To copy or move steps:

• From the PGM screen, highlight a step where you would like to add or insert the step(s) that you want to move or copy.

• Press the **STEP FUNCTIONS** softkey.

• Enter the first and last step numbers that you would like to move or copy.

• Press the **COPY/MOVE STEPS** softkey.

• Press either the **MOVE** or **COPY** softkey.

• Press the **USE** key.

Calculator

MILL*PWR*'s built-in calculator is capable of handling everything from simple arithmetic to complex trigonometry, geometry and RPM calculations. The numeric keypad resembles a standard calculator with keys for numbers 0 through 9, four math function symbols (+, - , x, and ÷), a decimal point, and a positive/negative sign (+/-).

The calculator is accessible from nearly any screen or field. Press the **CALC** key to access the "stand-alone" calculator and the **TRIG**, **GEOMETRY** and **RPM** softkeys. Calculations can be entered directly into a highlighted field.
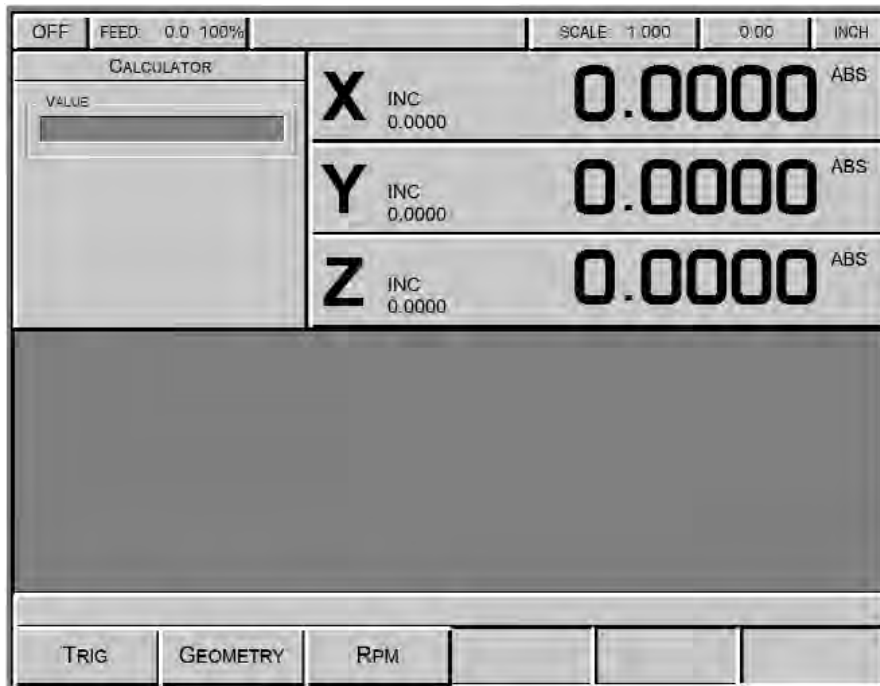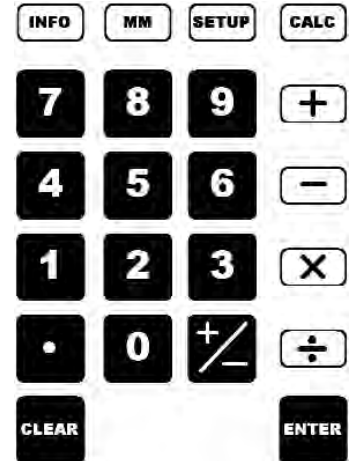
Math Functions (+, -, x, ÷)

Math functions may be performed separately in the stand-alone calculator or directly in the field you are working in. For example, let's say you enter the radius of a circle pocket, but the print only shows the diameter, 6.25. Here is an easy way to figure out the radius:

- Highlight the radius field for the circular pocket.

- Using the keypad, enter the value for the diameter, **6.25**.

RADIUS
6.2500 /

- Press the **÷** key.

- Press the **2** key.

- Press the **ENTER** key. The answer, 3.125, will appear in the "Radius" field.

When you need to enter more than one calculation into a numeric field, the calculator will perform multiplication and division before it performs addition and subtraction. If you were to enter 3 + 1 ÷ 8, **MILL***PWR* will divide one by eight, then add three for an answer of 3.125.

Trig Functions

As shown above, when entering data for a milling or positioning function, the print may not provide you with all of the information you need. If an unknown value can be expressed as a trigonometry function, square root or the square of a number, **MILL***PWR* can calculate the dimension for you in the trigonometry calculator.

Using the trig calculator:

- While in a numeric field (such as the "To" field for a Line), press the **CALC** key, and then press the **TRIG** softkey. The following softkeys will appear:

SHIFT	COS ARC COS	SIN ARC SIN	TAN ARC TAN	SQR ROOT SQR	( )

- Enter a number and then press the softkey with the appropriate trigonometry math function. (Press the **SHIFT** softkey to switch between the upper and lower functions.)

For example, to enter a radius that has a value equal to the square root of 2, follow this procedure:

- Highlight the radius field.

- Press the **2** key

- Press the **CALC** key

- Press the **TRIG** softkey

- Press the **SQR ROOT** softkey. A value of 1.4142 will appear.

When calculating the SIN, COS or TAN of an angle, enter the angle first and then press the appropriate softkey. If you were to enter a value of 30 into a numeric field, then press the **SIN** softkey, a value of 0.5000 (the SIN of 30 degrees) would appear.

Continuing with the example on the right, let's set up a 4" SIN plate at a 30° angle and need to know how tall the block underneath it should be. Multiplying 0.5000 by 4, shows we need to use a 2" block.

The parentheses softkeys can be used to group calculations within an arithmetic equation. For example,

2 + 1 ÷ 16 equals 2.0625, but (2 + 1) ÷ 16 equals 0.1875.

The parentheses are not always displayed during the keystroke sequence, but they are remembered by **MILL***PWR* as it calculates the correct answer.
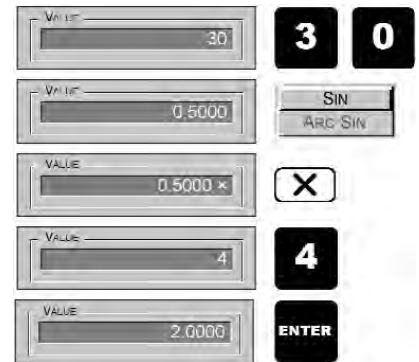
Geometry Functions

Working with the Geometry Calculator

To open the geometry calculator (GeoCalc), press the **CALC** key, and then press the "Geometry" softkey.
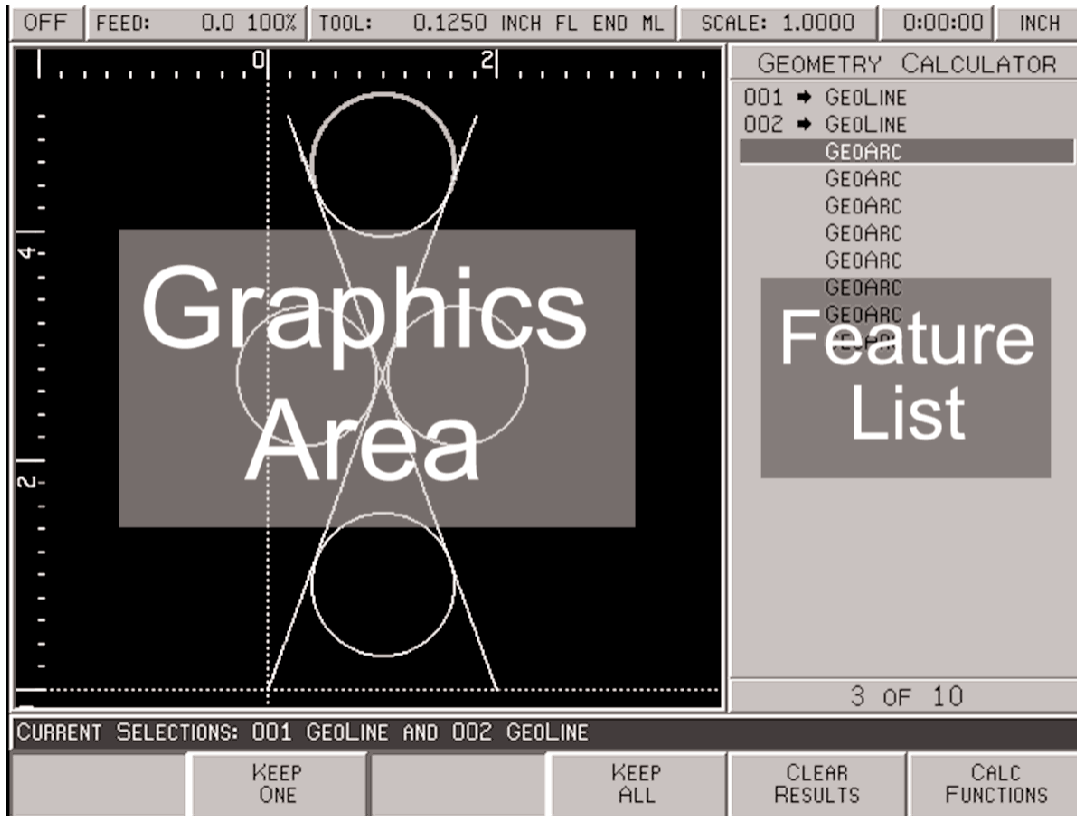
With **MILL*PWR***'s GeoCalc, you can calculate missing coordinates (such as the tangent point between a line and an arc) using the information that appears on theprint.

The kinds of calculations that GeoCalc performs depends on the items you select and whether you are trying to find a point, line, or arc. The table below lists the points, lines and arcs that can be found by GeoCalc.

Items Selected	Find Point	Find Line	Find Arc
2 Points	Midpoint between the given points	Line between the given points	Given a radius, all arcs through the given points The Auto Radius softkey gives you two arcs, each tangent to both points with the midpoint as its center point.
1 Point 1 Line	Point on the given line which, with the given point, would form a line perpendicular to the given line.	Lines through the given point, one parallel and one perpendicular to the given line.	Given a radius, all arcs through the given point and tangent to a given line. The Auto Radius softkey gives you two arcs, each tangent to the given line with the given point as its center point.
1 Point 1 Arc	Points which, with the given point, would form lines tangent to the given arc and the shortest line from the given point to the given arc.	Lines through the given point tangent to the given arc, and the shortest line through the given point to the given arc	Given a radius, all arcs through the given point and tangent to the given arc The Auto Radius softkey gives you all arcs tangent to the given arc with the given point as its center point.
1 Line	End points of the given line	Given a distance, lines parallel to the given line; lines perpendicular to the given line through its end points	
2 Lines	Intersection point of the given lines	Line that bisects the angle formed where the given lines intersect; line perpendicular to the bisector	Given a radius, all arcs tangent to both given lines
1 Line 1 Arc	Points where given line intersects given arc	Lines perpendicular to the given line and tangent to the given arc; shortest line to the given arc which is perpendicular to the given line	Given a radius, all arcs tangent to the given line and the given arc
1 Arc	Center point, end points	Given an angle, all lines tangent to and through the center point of the arc; lines from the end points to the center point	
1 Arc 2 Arcs	Intersection points	Lines tangent to arcs; shortest line between arcs	Given a radius, all arcs tangent to both arcs

Using GeoCalc:

- Press the **POS**, **LINE** or **ARC** key and enter the information that's on the print. (Refer to **Program Steps** for assistance.)

- To find a line, point or arc, highlight the feature(s) you wish to use, and then press the **SELECT FEATURE** softkey. An arrow will appear beside the feature(s) you have selected. (Only two features can be selected at one time.)

- Press the **FIND POINT**, **FIND LINE** or **FIND ARC** softkey—GeoCalc will automatically find all the possible points, lines or arcs for the features that have been selected.

SELECT FEATURE	FIND POINT	FIND LINE	FIND ARC	RETURN FEATURE	CALC FUNCTIONS

- Either highlight the feature you wish to keep and press the **KEEP ONE** softkey OR press the **KEEP ALL** softkey and retain all the features that are on the screen.

The **RETURN FEATURE** softkey transfers the object(s) that have been *highlighted* from GeoCalc and into the program or milling function, freeing you from having to write the information down and then entering it in number-by-number from the keypad.

Calculator Functions

GeoCalc has other useful features that make programming easier, like saving and loading calculations. Press the **CALC FUNCTIONS** softkey to access the following functions:

LOAD	SAVE	CLEAR	LOAD PROGRAM	USE FLOPPY	CALC FUNCTIONS

Saving Results Calculated in GeoCalc

GeoCalc lets you save your calculations on **MILL*PWR***'s internal memory or on a 3 $^1/_2$" floppy diskette so that you can retrieve them at a later time.

Note: You may only save the results of one GeoCalc session at a time onto **MILL*PWR***'s internal memory or onto a 3 $^1/_2$" floppy disk. Use one 3 $^1/_2$" floppy disk for each GeoCalc session you want to save.

To save the GeoCalc results onto **MILL*PWR***'s internal memory:

• From GeoCalc, press the **CALC FUNCTIONS** softkey.

• Press the **SAVE** softkey.

SELECT A CALCULATOR FUNCTION.

LOAD	SAVE	CLEAR	LOAD PROGRAM	USE FLOPPY	CALC FUNCTIONS

• To replace what you have previously saved, press the **YES** softkey. If you do not wish to proceed, press the **NO** softkey.

LOADING A SAVED CALCULATOR WILL CLEAR THE CURRENT CALCULATOR. PROCEED?

	YES			NO	

To save your GeoCalc results onto a 3 $\frac{1}{2}$" floppy disk:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.

- Insert a 3 $\frac{1}{2}$" floppy disk into the disk drive.

- Press the **USE FLOPPY** softkey.

- Press the **SAVE** softkey.

SELECT A CALCULATOR FUNCTION.					
LOAD	SAVE	CLEAR	LOAD PROGRAM	USE FLOPPY	CALC FUNCTIONS

If you have already saved the results from another GeoCalc session onto the 3 $\frac{1}{2}$" floppy disk, you will be asked if you want to replace what you previously saved. To save over the older session, press the **YES** softkey. If you don't want to save over the older session, press the **NO** softkey. Insert a new 3 $\frac{1}{2}$" floppy disk and repeat the procedure.

Loading GeoCalc Results That You Previously Saved

Once the results have been saved from GeoCalc, you can reload them into GeoCalc at any time.

To load GeoCalc results into GeoCalc from **MILL*PWR***'s internal memory:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.

- Press the **LOAD** softkey.

To load GeoCalc results into GeoCalc from a 3 $\frac{1}{2}$" floppy disk:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.

- Insert a 3 $\frac{1}{2}$" floppy disk into the floppy disk drive.

- Press the **USE FLOPPY** softkey.

- Press the **LOAD** softkey.

Loading Programs into GeoCalc

You can load the current or open program directly into GeoCalc. Because GeoCalc automatically recreates the program's features, you don't have to re-enter information.

To load the current program's features into GeoCalc:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.

- Press the **LOAD PROGRAM** softkey.

Clearing GeoCalc

Once you are finished working in GeoCalc, you can clear the screen.

To clear GeoCalc:

- From GeoCalc, press the **CALC FUNCTIONS** softkey.

- Press the **CLEAR** softkey.

- Press the **YES** softkey to clear the screen. If you do not wish to proceed, press the **NO** softkey.

Example Problem

Occasionally, you will have to use a print that doesn't provide all of the dimensions you need to program the part. In the illustration below for example, the coordinates of points C and D, where the arc is tangent to the sides, are unknown. These coordinates must be identified before programming lines B-C and A-D and the arc that extends from point D to point C.

Strategy

Program the line from A to B directly from the print. Then use GeoCalc to calculate the rest.

The strategy will be:

- Enter the line from A to D.

- Enter the line from B to C.

- Select lines A-D and B-C, and then use the **FIND ARC** function.

- Calculate points C and D, then return their coordinates and the arc into the program.
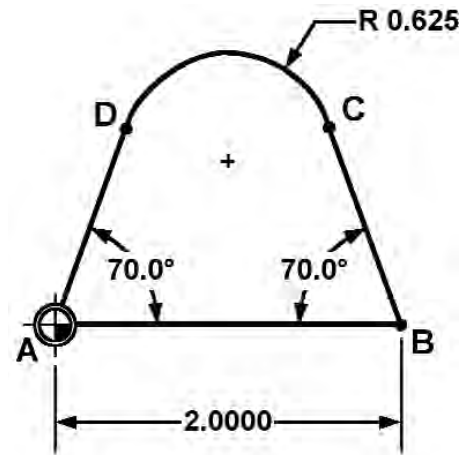
Starting the Program

In this example, point A is shown as datum. Its absolute coordinates are 0, 0. In the part program, draw a line by entering the absolute coordinates for point A (0, 0) in the "From" field and point B (2, 0) in the "To" field.

- Press **USE** to finish programming the line.
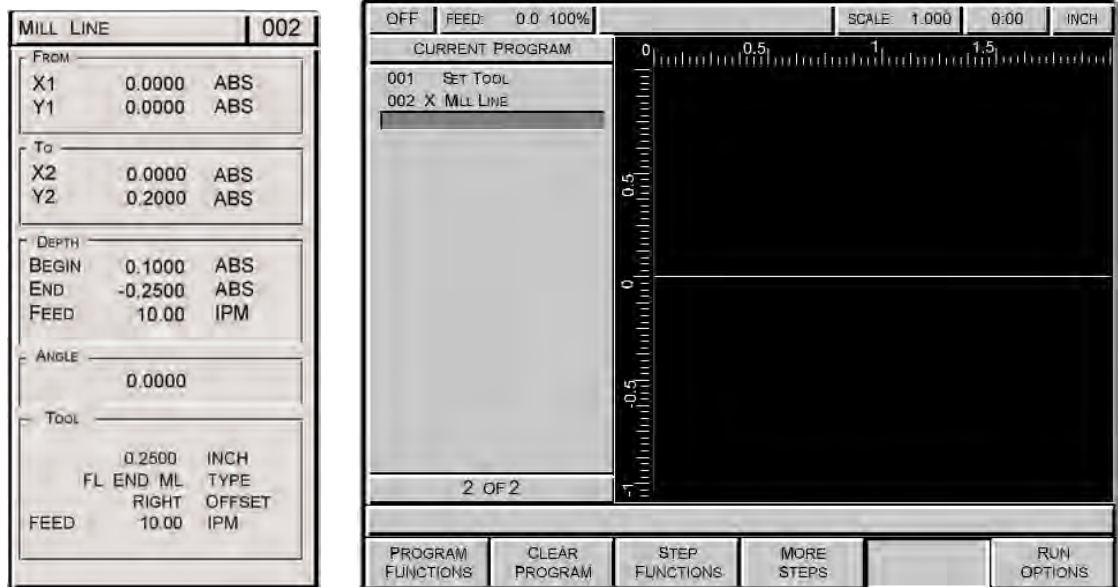
Now use GeoCalc to find the coordinates for points C and D.

Entering the Lines

Press the **CALC** key, and then press the **GEOMETRY** softkey. Now, press the **LINE** key and construct the line from A to D.

- Press the **LINE** key

- Enter the coordinates for
  the "From" point (point A):   X = 0, Y = 0

- The polar coordinates are R2 (radius) and A2 (angle). (Remember, a polar coordinate is a position defined by an angle and a radius.)

- If you don't know the length of the line, and GeoCalc doesn't need it. Just guess and enter a radius of 5".

- You know that the angle of the line from datum is 70°.

- Press the **USE** key, and a line starting at point A and traveling through point D will appear.
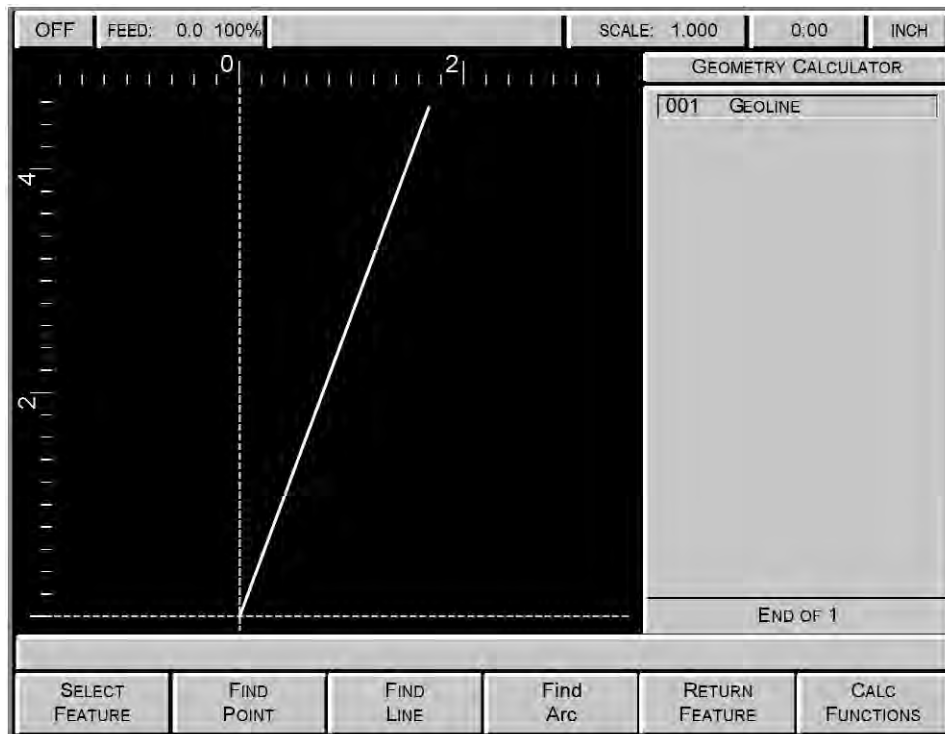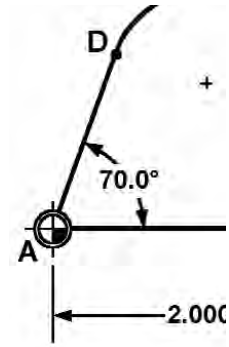
Next, enter the dimensions for the line that reaches from point B through point C.

- Press the **LINE** key.

- In the "From" field, enter point B's coordinates, X1 = 2 and Y1 = 0.

Now you are ready to enter the "To" point. Use polar coordinates, because you know the angle and can estimate the radius.

- Press the **POLAR** softkey. Guess "5" for the radius (R2).

- Press the **ABS/INCR** key to switch to incremental measuring (because we are measuring from point B, not from datum).

- Highlight the A2 field.

The print shows that the angle is 70°. But if 70° is entered, the line will slant the wrong way. The complementary angle is needed instead. You can figure it out by subtracting 70° from 180°, the total angle of the line.

- You can either enter the answer, 110°, directly into the (A2) field, or you can calculate the answer using the keypad ("180-70").

- Since the angle is measured from point B (incremental zero) you must press the **ABS/INCR** key again.

- Press the **USE** key, and line B-C will appear.
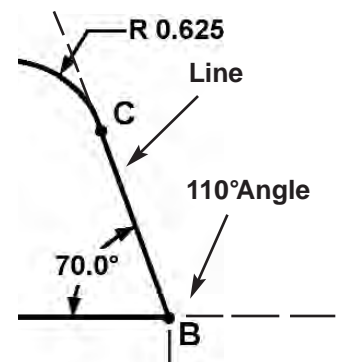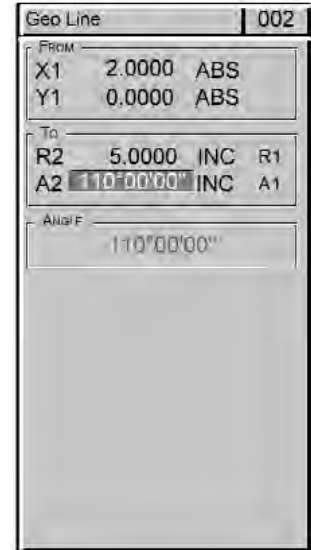
Finding the Arc

Now that the lines are complete, it's time to find the arc. To do this by select the lines already drawn and then use GeoCalc to find the arc.

- Highlight 001 GEOLINE and press the **SELECT FEATURE** softkey. The arrow indicates that the feature has been selected.

- Now highlight 002 GEOLINE and press the **SELECT FEATURE** softkey.

- Press the **FIND ARC** softkey. The message bar will display a field for the arc radius.

- The print indicates that the arc's radius is 0.625. Using the keypad, enter "0.625," and then press the **USE** key.
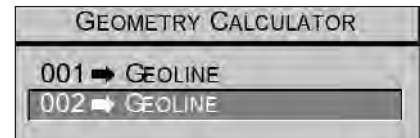
GeoCalc will display all of the arcs that have a 0.625 radius and are tangent with the two lines you've selected (A-D and B-C).

- Using the arrow keys, scroll through the list of found GeoArcs until the arc you want to keep is highlighted.

- Press the **KEEP ONE** softkey, and GeoCalc will save the arc as item 003.

- Arrow down and then press the **CLEAR RESULTS** softkey to erase the remaining arcs.
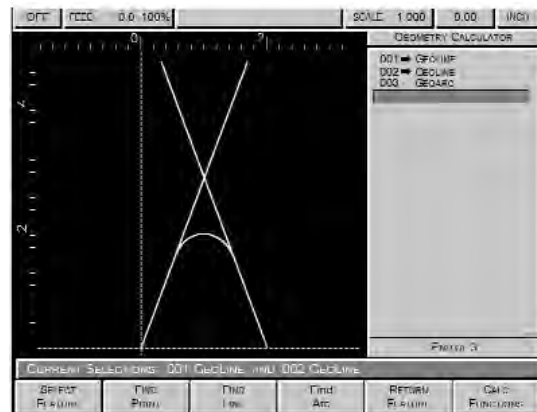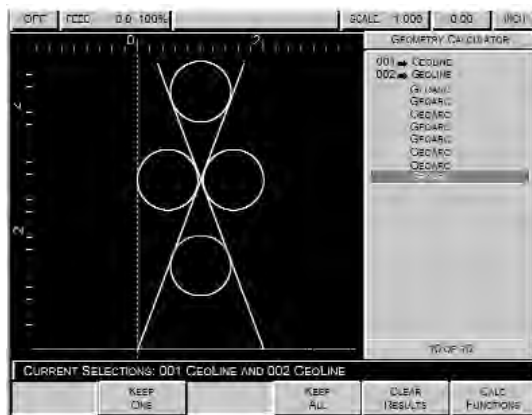
Finding the Points of Tangency

The points of tangency between the lines and the arc need to be determined (the coordinates for points C and D). Now that the arc is in place, the coordinates of its end points are needed. Use GeoCalc's "Find Point" feature to find the end points of the arc.

- Highlight 001 GEOLINE and press the **SELECT FEATURE** softkey. This will de-select the line and cause the arrow to disappear.

- Highlight 002 GEOLINE and press the **SELECT FEATURE** softkey to de-select it.

- Highlight 003 GEOARC and press the **SELECT FEATURE** softkey to *select* it.

- Press the **FIND POINT** softkey. Three points will appear—one is the center of the arc and the others are the ends of the arc. The point on the right is the one needed; it's point C on the drawing.

- Using the arrow key, scroll down through the list of "GeoPoints" until the point on the right is highlighted. Press the **KEEP ONE** softkey.

The tangent point (C) is now stored in the GeoCalc as item 004.
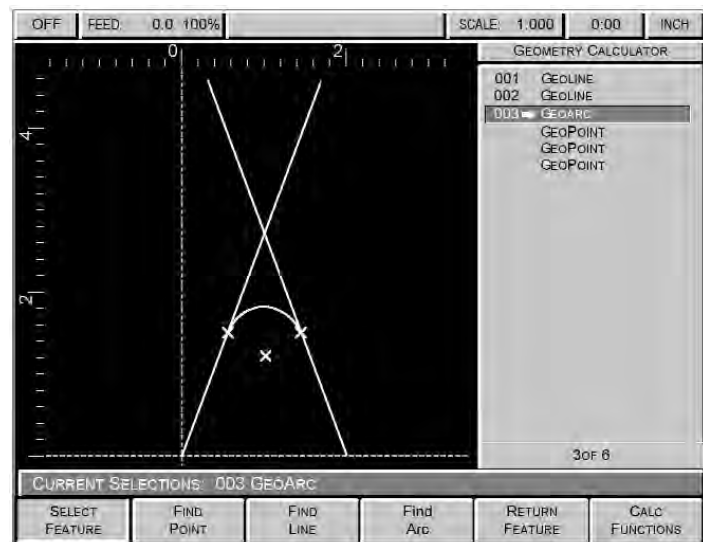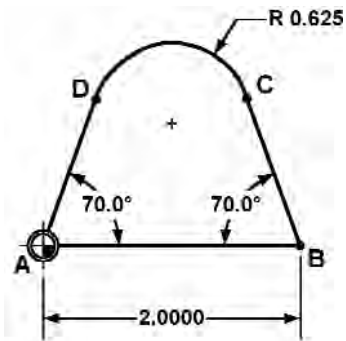
Returning Features

Now bring the GeoCalc results back into the program. When you press the **RETURN FEATURE** softkey, GeoCalc will transfer the feature you have identified into the part program.

- Press the **CANCEL** key to return to the program. Check that the cursor is just *below* the line previously programmed (item 001).

- Press the **LINE** key to begin a new "Mill Line" form. The "From" point is automatically set at point B. The "X2" field should be highlighted. This is point C, one of the points found in GeoCalc.

- Press the **CALC** key, then the **GEOMETRY** softkey.

- Highlight the GeoPoint representing point C (step 004). Press the **RETURN FEATURE** softkey. The program screen will return and point C's coordinates will appear in the "To" field.

- Press the **USE** key to accept the line into the part program.

- Press the **CALC** key and then the **GEOMETRY** softkey.
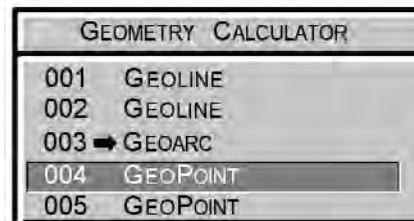
The features in the calculator will not be lost until the MILLPWR is turned off or until you clear the calculator.

- Highlight the arc.

- Press the **RETURN FEATURE** softkey. **MILL*PWR*** will automatically add an arc step to the program and place the arc's information into the form.

- Press the **USE** key to accept the arc step into the part program.

- Press the **LINE** key. This line ends at datum, so enter "X2 = 0" and "Y2 = 0" for the "To" point, and press the **USE** key to keep the line.
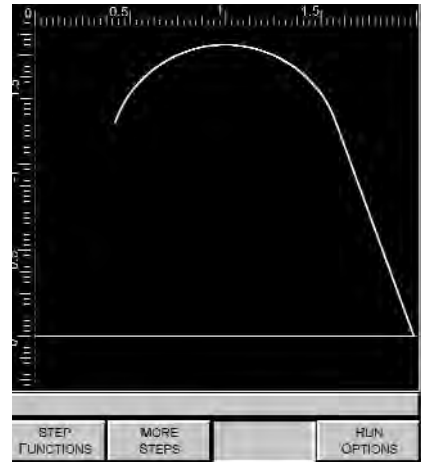
The contour is now complete.
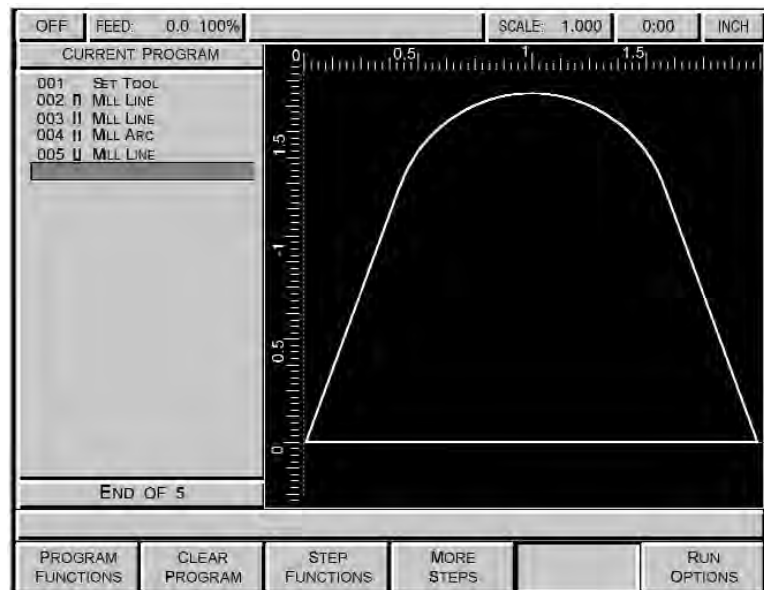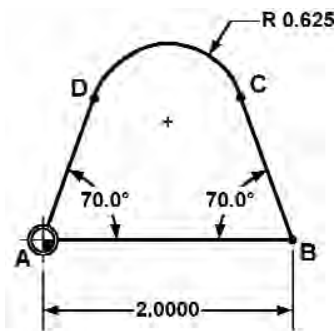
RPM Functions

MILL*PWR* includes a built-in RPM calculator that allows you to make RPM calculations in seconds. It calculates the RPM based upon the tool diameter and surface speed.

If the cursor is in the RPM field of a "Set Tool" step before you open the RPM calculator, **MILL***PWR* can transfer the result directly into the field for you.

To use the RPM calculator:

- Press the **CALC** key, then the **RPM** softkey.

- Enter the diameter of your tool.

- Enter the surface speed for the material and cutter type.

  Press the **SURFACE SPEED TABLE** softkey to view a list of recommended surface speeds.
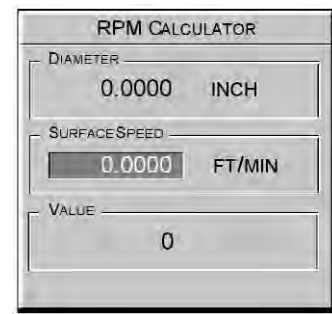
- If you need to switch from feet per minute to meters per minute, or vice versa, use the **FEET/MIN** and **METERS/MIN** softkeys.

- Press the **ENTER** key. **MILL***PWR* will calculate the RPM value.

- If the cursor was in the RPM field of a set tool step before you accessed the RPM calculator, press the **USE** key and **MILL***PWR* will transfer the result directly into the "RPM" field for you.

Setup

Press the **SETUP** key to access the setup parameters. The setup parameters are grouped into two categories: Job Setup and Installation Setup.

Job Setup parameters can easily be changed to accommodate specific machining requirements for each job.

Installation Setup parameters, on the other hand, are established during the initial installation and, with few exceptions, should not be changed. For this reason, the installation setup parameters are protected by a passcode. The passcode is located on the page before the table of contents. You can remove it and store it in a separate location to prevent accidental or unauthorized adjustments to the system.

OFF	FEED:	0.0 100%			SCALE: 1.000	0:00	INCH

JOB SETUP

TOOL LIBRARY
SCALE FACTOR
DISPLAY OPTIONS
ELECTRONIC EDGE FINDER
JOB CLOCK
FEED RATE SETTINGS
INSTALLATION SETUP

1 OF 7

				USE NEW SETTINGS	CANCEL CHANGES
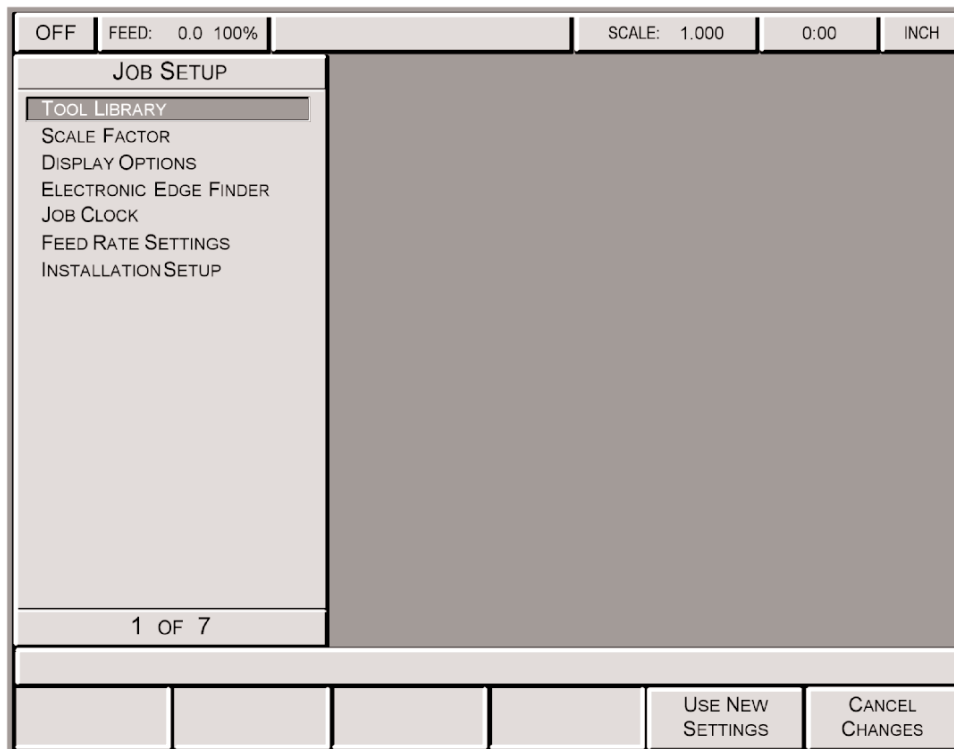
To change the setup parameters:

- From Setup, highlight a topic, then press **ENTER**. Change the settings, then press the **USE NEW SETTINGS** softkey to activate them. (Or press the **CANCEL CHANGES** soft-key to exit Setup and return to the previous screen without making any changes.)

Job Setup

Tool Library

MILL*PWR* contains a tool library that can store the diameter, length, unit of measure, and tool type for up to 99 tools. **MILL*PWR*** only requires that you provide the tool's diameter; however, it's a good idea to provide additional information so that each tool is easier to identify from the tool list.

Set up the tool library so that the tools you use most often appear first.

You can sort the tool library by number or type.

⚠ Pressing the **RESET** softkey will clear all of the tools that are programmed in the tool library. Only use this key when you want to start over.

OFF	FEED:	0.0 100%	SCALE: 1.000	0:00	INCH

TOOL LIBRARY

1	0.2500	/	1.4756 INCH	FLAT END MILL
2	0.5000	/	2.9875 INCH	FLAT END MILL
3	1.0000	/	3.5575 INCH	ROUGHING MILL
4	6.0000	/	0.0000 INCH	FLY CUTTER
5	0.4219	/	0.0000 INCH	PILOT DRILL
6	0.2500	/	0.0000 INCH	DRILL
7	0.3750	/	2.5875 INCH	COUNTER BORE
8	0.2500	/	0.0000 INCH	TAP
9	5.0000	/	17.0000 MM	DRILL
10				
11	0.3750	/	0.0000	TAP
12				
13				
14				
15				
16				
17				
18				
19				

1 OF 99

| SORT BY NUMBER | SORT BY TYPE | RESET | | USE NEW SETTINGS | CANCEL CHANGES |

Using the tool library without repeatable tool lengths:

To enter a tool into the Tool Library:

- From Setup, highlight "Tool Library" then press **ENTER**.

- Highlight a blank row in the Tool Library and press **ENTER**.

- Enter the tool's diameter.

- Highlight the "Type" field.

- Press the **TOOL TYPES** softkey.

- Highlight the tool type that describes the tool being used. Press **ENTER**.

- Press the **USE** key.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

To edit an existing tool:

- From Setup, highlight "Tool Library" then press **ENTER**.

- Highlight the tool you want to edit, then press **ENTER**.

- Make the necessary changes, then press **USE**.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

If you need to, change the units (inches or mm) for the tool you are using before you set the length.

TOOL DESCRIPTION

DIAMETER

LENGTH
0.0000

UNIT
INCH

TYPE

Using the Tool Library with Repeatable Tool Length Offsets

If you have repeatable tooling, keeping a tool library for frequently used tools, saves you time by not requiring you to enter the tool information every time you program the tool.

When running G-code programming, the tool length for each tool in the program is retrieved from the library. For example, in a G-code program, "T1 M6" will cause **MILL*PWR*** to prompt the operator to load tool 1 into the spindle. The tool length off set from tool #1 in the **MILL*PWR*** tool library is retrieved and used to adjust the Z-axis position. The code T2 uses the length offset of tool 2, etc. This feature takes advantage of the repeatable tooling systems, freeing the operator from having to reset the Z-datum for each tool change.

If the Tooling is not repeatable, you should not place any length information into the tool library.
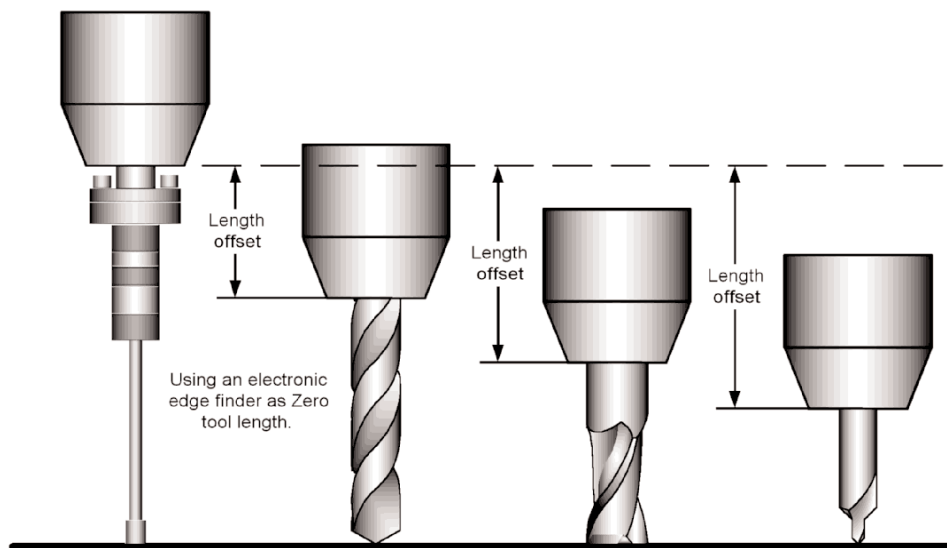
All tool lengths originate from datum. There are several different ways to establish datum, such as "touching off" with an empty spindle, using a repeatable standard or using an electronic edge finder (refer to **Establishing Datum**).

If you begin setting the tool length offsets by setting Datum using an electronic edge finder it must have a fixed and repeatable length. The current tool information can not have a length value. All of the tool length offsets in the program will be the difference in length between the tool and the edge finder.

When defining tool length offsets in the Tool Step, touch each tool to the same surface and press the **TEACH TOOL LENGTH** softkey to establish the tool length.

Once you have identified the tools you will be using, you can program the length offsets into the tool steps.

Entering the first tool

Decide on a method that will be used to set the Z-axis datum. You can choose to set the Z-axis datum with the spindle nose (without a tool inserted), using a repeatable standard, or using a 3-D electronic edge finder. To begin your Tool Library, set the current tool information in the status bar so the tool length is zero. In this example, you will begin with the spindle nose to set datum zero. If you wish to use the ACU-RITE 3-D Electronic Edge Finder, or a tool holder with a repeatable standard, then place it into the spindle instead.

- Remove any tool from the spindle.
- From the DRO screen, press the **TOOL** key.
- Enter zero for the diameter and zero for the tool length.
- Press **GO**.
- An Operator intervention message will appear asking you to "use" the entered tool.
- Press **GO** again.
- **MILL*PWR*** is now set with a zero tool length offset.
- Press the **DATUM** softkey.
- Touch the spindle nose to a solid surface such as the top of a vice or the workpiece.
- Press the **Z=0** softkey, then the **USE** key.

Datum zero for the Z-axis is now set to the surface that the spindle nose touched. Use this same location to "teach" each tool length in the tool library.

- Press the **SETUP** key and highlight "Tool Library."
- Press **ENTER**.
- Highlight the tool number in the tool library list, for the tool you are setting up and press **ENTER**.
- Use the numeric keypad to enter the tool's diameter. If the diameter you entered is metric, press the **MM** softkey.
- Highlight the length field.
- Place this tool into the spindle and touch its tip to the same surface you used to set the Z-axis datum Zero.
- Press the **TEACH TOOL LENGTH** softkey. The length offset for this tool will appear in the length field.
- Highlight the "Type" field.
- Press the **TOOL TYPES** softkey.
- Highlight the type of tool from the list of available tool types and press **ENTER**.
- Press **USE**.
- Repeat this procedure for all additional tools then, press **USE NEW SETTINGS** to complete the setup.

After the tool information is established in the library, you can edit the values as needed to complete the setup.

Using the Tool Library

Whenever you program a tool step, you will see a **TOOL LIBRARY** softkey. Press the **TOOL LIBRARY** softkey and highlight the tool you wish to use. Press **ENTER**, then **USE**, and the tool step is programmed. It is important to understand that the tool information that is retrieved from the library remains in the program.

If you change a tool in the tool library, the tool information in all programs will not change.

The program will continue to prompt for the tools you originally programmed. If you wish to update a tool step in a program, highlight the tool step you wish to change, press **ENTER**, press the **TOOL LIBRARY** softkey and highlight the tool you wish to use. Press **ENTER**, then **USE**, and the tool step is updated for the new tool.

Setting Datum

Once a library of tools has been established, use any one of the known tools to establish Datum. To establish Datum with a tool from the tool library.

- From the DRO view, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list and press enter.
- Press the **GO** key. An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that the tool has been installed.
- Follow the instructions for setting datum in the operation manual.

Once Z-axis datum is set with a known tool from the tool library, all of the tools in the tool library can be used without the need to reset datum.

It is extremely important to set Z-axis datum on a solid surface with a tool from the library after running a tool step with this tool, or by using the identical method used when setting the first tool, before you input additional tools to the library. If this is not done, the tool length cannot be determined correctly.

Changing between Tools in the Tool Library when in DRO view

To change tools when in the DRO view:

- From the DRO view, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list and press enter.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that the tool has been installed.

Changing to a Tool of unknown length when in DRO view

- From the DRO view, press the **TOOL** key.
- Enter the tool diameter and if you wish, the type.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool in the spindle.
- Press the **GO** key to confirm that the tool has been installed.
- Set the Z-axis datum following the instructions found in the operation manual.

At this point, the tool library length offsets are not in effect. Reestablishing the Z-axis datum with a tool from the tool library will reestablish the tool length offsets.

- From the DRO view, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list and press enter.
- Press the **GO** key.
- An Operator intervention message will appear asking you to "use" the selected tool.
- Place the selected tool into the spindle.
- Press the **GO** key to confirm that the tool has been installed.
- Reset the Z-axis datum following the instructions found in the manual.

Using the tool library within a program

- When programming, press the **TOOL** key.
- Press the **TOOL LIBRARY** softkey.
- Highlight the tool you would like to use from the tool library list.
- Press the **ENTER** key, then the **USE** key.

When the program is executed and the tool step is reached, an Operator intervention message will appear asking you to "use" the programmed tool.

- Place the selected tool in the spindle. A **MOVE TABLE** softkey is available to help you move away from the workpiece if necessary. You can program a position move prior to the tool step to automatically move off the part before changing the tool.

- Press the **GO** key to confirm that you have installed the tool. The program will resume.

Changing to a Tool of unknown, but repeatable length when executing a program

When running a program, it is easy to use tools of unknown but repeatable lengths, such as a counter bore placed in an end mill holder. As long as the counter bore does not leave the end mill holder, its length will repeat each time it is placed in the spindle. Since the tool length remains the same, you can teach this length to **MILL*PWR*** when creating the program without having to enter the tool into the Tool Library.

- Be sure the Z-axis datum was set using a tool from the library (See: Using the Tool Library above).
- When programming the tool step, enter the tool diameter, then place the tool in the spindle and touch Z-datum.
- Highlight the length field and press the **TEACH TOOL LENGTH** softkey. The tool length offset will appear in the length field.
- Enter any of the remaining information you need in the tool step and press **USE**.
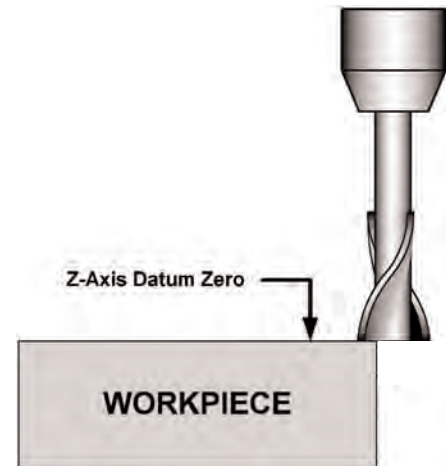
When this tool step is executed, the length offset will take effect and the tool tip will be offset correctly.

Z-Axis Datum Zero →

WORKPIECE

Changing to a Tool of unknown, non-repeatable length when executing a program

When running a program, it is possible to switch between tools of known length from the library, to tools that have a non-repeatable length such as a counter-bore held in an R-8 collet. Follow this program as an example:

Step 1 is a tool step, tool 1-a center drill-programmed from the tool library and is repeatable.
Step 2 in the program is a bolthole pattern.
Step 3 is a tool step using tool 2-a drill-from the library and is also repeatable.
Step 4 is the same hole pattern repeated.
Step 5 is a tool step programmed as a counter bore of unknown length and does not repeat.
Step 6 is the same hole pattern repeated.

For this example, two identical parts are being machined. The first part is in place and datum has been set using tool 1 from the library as the current tool. The top of the work-piece is Z-zero. When the **GO** button is pressed, **MILL*PWR*** will see that step one is asking for the same tool as the current tool and will proceed to the next step (Step 2) and drill the holes. When the next tool step (Step 3) is executed, you will be prompted to change tools to the drill. Simply change the tool and press **GO** to continue. The programmed tool offset will compensate for the difference in tool length as it drills the holes (Step 4). The next tool (Step 5) is of unknown length. When this step is executed, and you are prompted to change tools. With this tool change:

- Press the **DATUM** softkey. The DRO screen will appear with the set datum window.
- Place the tool (counter bore) in the spindle and touch the cutting edge to Z-axis Datum Zero (the top of the workpiece in this example).
- Press the **Z=0** softkey and then **USE**. The tip of the tool is now set at datum zero.
- Press GO to continue with the program.

Now that the Z-axis datum was set using a tool of unknown length, tools one and two will not repeat. The Z-axis datum must be reset once again using a tool from the library. This can be done when step 1 is run on the second part. After the part has been changed:

- Press GO. You will be prompted to change tools to the center drill (Step 1).
- Press the **DATUM** softkey. The DRO screen will appear with the set datum window.
- Place the center drill in the spindle and touch the tip to Z-axis Datum Zero (the top of the workpiece in this example).
- Press the **Z=0** softkey and then **USE**. The tip of the tool is now set at datum zero.
- Press GO to continue with the program.

Datum has now been reset with a repeatable tool with a programmed length offset as the current tool, all of the tools with programmed length offsets again relate to datum. The program can continue as usual.

Note: The only information that is mandatory when entering a tool into the tool library, is the diameter. All other information is optional. It is possible to establish a tool library without tool lengths. Z-axis datum will need to be set during each tool step execution of the program.

Scale Factor

Scale factor is simply a multiplier that lets you expand or shrink the print's dimensions without per forming complex calculations or affecting the proportion of the part. Simply enter the number that you would like the dimensions to be multiplied by, then run the program as you normally would. **MILL*PWR*** will adjust the programmed dimensions (but not the tool size) automatically.

MILL*PWR*'s scale default setting is 1.0000. A value greater than 1.0000 will increase the part's dimensions; a value less than 1.0000 will reduce them.

MILL*PWR* will display the current scale factor in the status bar (located toward the top of the display screen).

To change the scale factor

- From Setup, highlight "Scale Factor," then press the **ENTER** key.

- Make the necessary changes, then press the **USE** key.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.
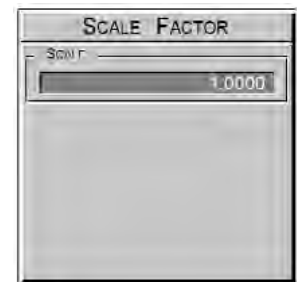
Display Options

From the "Setup" menu, select "Display Options" to customize the default settings for display resolution, point entry and similar items.

**ANGLES** allows you to define how you would like to enter angle values—as decimal degrees, radians or degrees–minutes–seconds (DMS). Select the angle default setting that satisfies your need most often. In other words, if the angles are usually dimensioned in decimal degrees, you should make **DECIMAL DEGREES** the default setting.

INC DISPLAY enables you to define how you want your incremental DRO displays and the "From" and "To" fields for milling functions to be displayed. Select either "Incremental Travel" or "Distance to Go."

- INCREMENTAL TRAVEL- In this view, the dimensions you program will end at their programmed distance. If you program a 1/4" move, the incremental display will read 0.2500" when you reach the end of the move.

- DISTANCE TO GO- In this view, the dimensions you program will be "preset" into the readout display. Every move will end at zero. If you program a 1/4" move, the incremental display will read 0.0000" when you reach the end of the move.
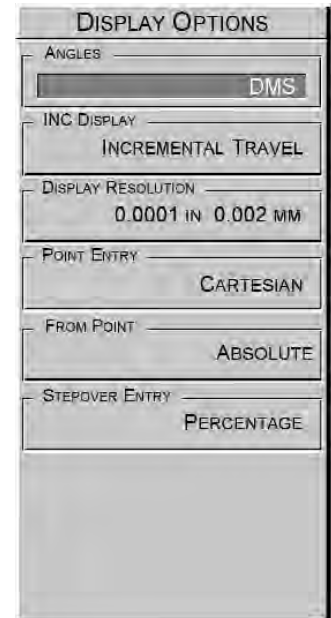
DISPLAY RESOLUTION lets you select the resolution you want to display in the DRO.
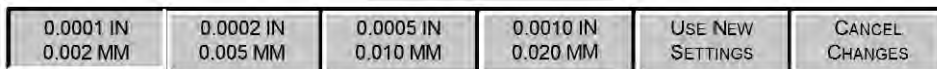
POINT ENTRY lets you decide what the default setting will be for entering dimensions for both "From" and "To" points—as either Cartesian (X and Y) or Polar (radius and diameter) coordinates. The method that you do not select will be available as a softkey when you are defining "From" and "To" points.

FROM POINT allows you to determine whether the "From" point will appear as an absolute value or an incremental value. Absolute values are based upon the distance from datum. Incremental values are based upon the distance from one feature or step in a program to another. No matter which you choose as a default setting, the other will always be available by pressing the **ABS/INCR** key.

Absolute "From" points will appear like this:

    MILL LINE   002
    From
    X1   4.0050 ABS
    Y1   2.5025 ABS

Incremental "From" points will appear like this:

    MILL LINE   003
    From
    X1   0.0000 INC   002
    Y1   0.0000 INC   002

STEPOVER ENTRY enables you to define how much material the tool will remove as it cuts a pocket, slot or ring. To enter a value as a decimal, press the **DECIMAL** softkey. To enter a value using whole numbers, press the **PERCENTAGE** softkey.

Electronic Edge Finder

An electronic edge finder enables you to "teach" positions, find the center point of a circle, skew a part or locate datum (also known as workpiece zero) by simply "touching off" on the part. The greatest advantage of an electronic edge finder is that it instantly senses when contact has been made —even if you have over-traveled.

ELECTRONIC EDGE FINDER
DIAMETER
0.2000
UNIT
INCH

MILL*PWR* lets you define the "Diameter" and "Unit" of measure (either inches or millimeters) for an electronic edge finder. Once this information has been entered into **MILL*PWR***, **MILL*PWR*** will automatically compensate for the radius of the tip of the electronic edge finder when performing any of the operations mentioned above.
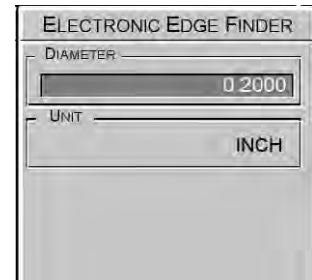
> <u>To define the diameter and unit of measure for an electronic edge finder:</u>
>
> • From Setup, highlight "Electronic Edge Finder" then press the **ENTER** key.
>
> • Enter a value for the electronic edge finder's tip diameter and unit of measure (either inches or millimeters) then press the **USE** key.
>
> • Continue with Setup or press the **USE NEW SETTINGS** softkey.

Job Clock

MILL*PWR* includes a job clock that can be used to keep track of the time you spend on each job. The job clock works like a stopwatch, counting in one minute intervals after you have pressed the **START** softkey and stopping after you have pressed the **STOP** softkey.

The job clock appears in the "Status" bar and is visible from both the PGM and DRO screens. It reflects how much time has elapsed since the clock was started. Every time you stop the job clock, the time will be added to the "Elapsed Time" field on the "Job Clock" menu and the value in the "Status" bar will be reset to zero. You can use the job clock to time each step of a project and then refer to the "Elapsed Time" for the total time spent on the job.
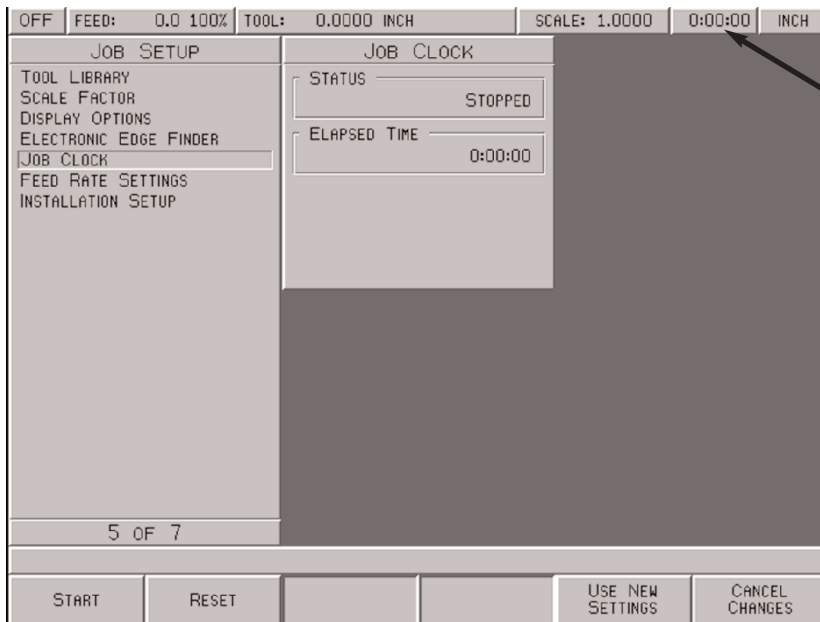
Job Clock

Note: The maximum amount of time that the job clock will record is 99 hours, 59 minutes and 59 seconds. The value will then roll over to zero and continue counting.

To start the job clock:

- From Setup, highlight "Job Clock" then press the **ENTER** key.

- Press the **START** softkey, and the job clock will begin.

To stop the job clock:

- From Setup, highlight "Job Clock" then press the **ENTER** key.

- Press the **STOP** softkey, and the job clock will stop. The job clock's current time will be added to the value in the "Elapsed Time" field.

- To restart the job clock, press the **START** softkey.

To reset the job clock and the "Elapsed Time" field:

- From Setup, highlight "Job Clock" then press the **ENTER** key.

- Press the **RESET** softkey, and the job clock and "Elapsed Time" field will be cleared.

Feed Rate Settings

Here you can change the default settings for various speeds and feeds:

STEP OVERRIDE % determines the percentage of change that occurs when you press the **FEED+** and **FEED-** keys.

MAX % establishes the maximum feed rate percentage that the **FEED+** key will allow.

MIN % establishes the lowest feed rate percentage that the **FEED-** key will allow.
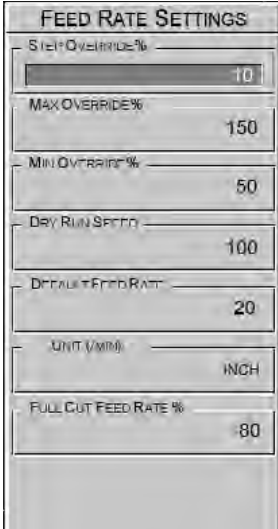
DRY RUN SPEED is the speed that the table will move during a dry run.

DEFAULT FEED RATE enables you to define the default feed rate for all milling functions—this way, if you forget to enter a feed rate,
MILL*PWR* will automatically apply the default feed rate to the milling function. It is suggested that you set the default feed rate to the rate that you use most often.

UNIT/(MIN) lets you define the feed rate in either inches per minute or millimeters per minute.

FULL CUT FEED RATE % is the percentage of the programmed feed rate used when the tool is making a full cut (e.g., the beginning cut in a pocket).

Installation Setup

After selecting "Installation Setup" from the "Job Setup" list, you will be asked for a passcode. The passcode is located on the page before the table of contents. You can remove it and store it in a separate location to prevent accidental or unauthorized adjustments to the system. Once you've entered the passcode, the "Installation Setup" screen will appear.

```
OFF | FEED:    0.0 100% |                    | SCALE: 1.0000 | 0:00:00 | INCH
     INSTALLATION SETUP
   PROTECTION
   ERROR COMPENSATION
   ENCODER SETUP
   TRAVEL LIMITS
   ERROR CHECKING
   SERIAL PORT
   SPINDLE CONTROL
   ERROR LOG
   SERVO TUNING
   AMI OUTPUTS
   DIAGNOSTICS
   MAX SERVO SPEED


        1 OF 12

  PREVIOUS                                        USE NEW      CANCEL
   MENU                                           SETTINGS     CHANGES
```
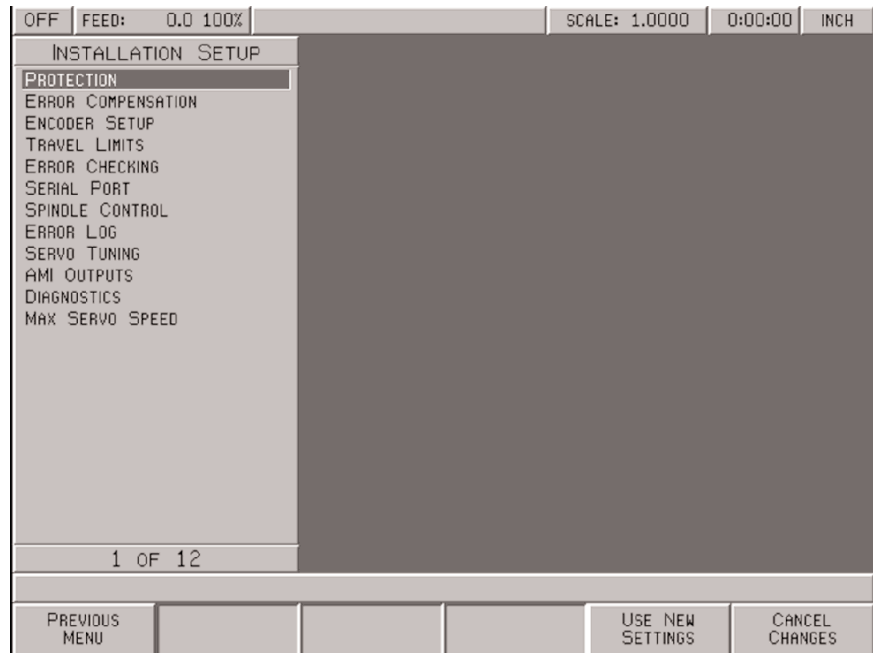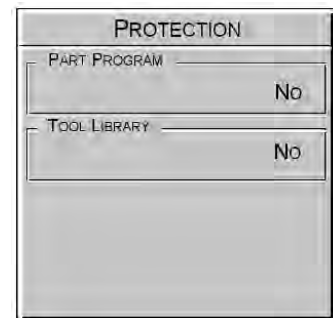
Protection

Protection enables you to prevent unauthorized access to your part programs and tool library. With "Part Program" and "Tool Library" set to "Yes," the part programs and tool library cannot be altered—without first changing the "Part Program" and/or "Tool Library"setting's to "No."

```
        PROTECTION
  PART PROGRAM
                          No
  TOOL LIBRARY
                          No
```

> To change the protection settings:

- From Installation Setup, highlight **PROTECTION** then press **ENTER**.

- Use the **NO** and **YES** softkeys to disable or enable protection for both the "Part Program" and "Tool Library."

- Press the **USE** key.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Error Compensation

Before performing linear error, compensation, you need to ensure that the machine is properly adjusted. Refer to the manufacturers' instructions.

MILL*PWR* is able to automatically calculate and compensate for machine tool wear.

If you know the error compensation value in parts per million (PPM), you can enter it directly. Otherwise, use the following procedure to determine the value for each axis:

Note: You can only perform error compensation for one axis at a time.

- Set up a standard of known length (e.g., a gage block), by indicating it in along the axis to be measured. Check that the standard is straight and parallel with the axis you are working with.

- Enter the length of the standard.

  If you need to touch each end of the standard using opposite edges of the electronic edge finder or mechanical indicator, be sure to include its diameter when entering the length of the standard. (Refer to the graphic below.)

Check between the first and second points several times— note the error value to ensure repeatability.

- Touch one end of the standard and press the **FIRST POINT** softkey.

  If you are using an electronic edge finder, you won't have to press the first point softkey.
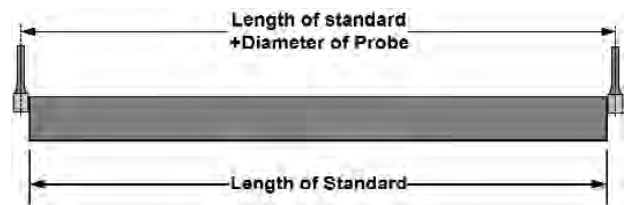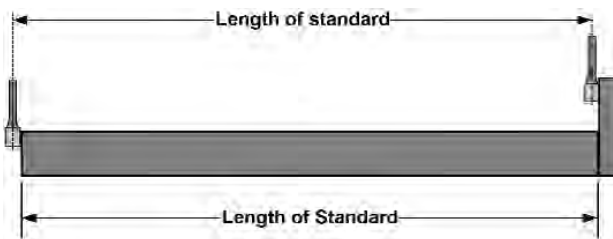
- Touch the opposite end of the standard and press the **SECOND POINT** softkey. If you are using an electronic edge finder, you won't have to press the second point softkey.

  MILL*PWR* will determine and automatically enter the correct compensation factor in parts per million.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Encoder Setup

When this **MILL*PWR*** system was installed, the encoder count direction should have been established for each axis. Each scale should have been assigned to move in either a positive or negative count direction, depending on how and where the scales were mounted to the machine.

> ⚠ You shouldn't have to change your X- and Y-axes encoder directions. Should you want the count directions changed, please contact your ACU-RITE distributor.

You *can* change the count direction for the Z- and W-axes. The Z-axis measures the movement along the quill, and the W-axis measures movement along the knee.

With **MILL*PWR's*** coupling feature, you can add, subtract, or average the movements of two parallel axes—in this case the Z- and W-axes. Coupling the Z-axis with the W-axis allows you to maintain your Z-axis datum and tool offsets whether you move the Z-axis or the W-axis. When the axes are coupled, both change the Z-axis DRO count value.

Be careful about changing the count direction and then running an old program—you could get some pretty strange results!

To change the encoder direction and coupling settings:

- From Installation Setup, highlight "Encoder Setup" and press **ENTER**.

- Highlight the axis you wish to change. Now use the **NEGATIVE** and **POSITIVE** softkeys to change the count direction.

- Highlight "Coupling," and use the **OFF**, **COUPLE Z+W** and **COUPLE Z-W** softkeys to change coupling.

- Press the **USE** key.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Travel Limits

Travel limits define how far you will allow the table to move along the X- and Y-axes.

Note: You must find home before you can establish the travel limits.

To set your travel limits:

- From Installation Setup, highlight "Travel Limits" and press **ENTER**. The "X-axis Travel" field should be high-lighted.

- Move the table to the position where you want the right table travel limit set.

- Press the **SET RIGHT LIMIT** softkey.

- Move the table to the position where you want the left table travel limit set.

- Press the **SET LEFT LIMIT** softkey.

    Note: The X-axis travel field will update to show you the maximum allowable traveling distance between the left and right table travel limits.

- Now highlight the "Y-axis Travel" field.

- Move the table away from you until you have reached the position where you want the inside table travel limit set.

- Press the **SET INSIDE LIMIT** softkey.

- Move the table towards you until you have reached the position where you want the outside table travel limit set.
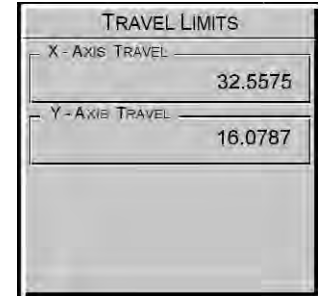
Use the handles or the MOVE TABLE feature to position the table.

If you use MOVE TABLE, keep in mind that the table travel limits are not activated.

TRAVEL LIMITS

X - AXIS TRAVEL
32.5575

Y - AXIS TRAVEL
16.0787

- Press the **SET OUTSIDE LIMIT** softkey.

  Note: The Y-axis travel field will update to show you the maximum allowable travel distance between the inside and outside table travel limits.

- Press the **USE** key.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

You may change the table travel limits at any time.

Error Checking

With the error checking feature enabled, **MILL*PWR*** can detect scale miscounts and record them in the Error Log. **MILL*PWR*** will stop the program if a scale miscount is detected.

To enable/disable the error checking feature:

- From Installation Setup, highlight "Error Checking" and press **ENTER**.

- Press either the **ENABLE** or **DISABLE** softkey.

- Press the **USE** key.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

ERROR CHECKING
X AXIS — ENABLED
Y AXIS — DISABLED
Z AXIS — DISABLED
W AXIS — DISABLED

Serial Port

You can connect **MILL*PWR*** to a PC via the serial port (located on the rear of the operator console). The serial port allows you to store part programs on a PC.

To activate the serial port:

- From Installation Setup, highlight "Serial Port" and press **ENTER**.

- Press the **REMOTE STORAGE** softkey to change the "Function" setting from "Off" to "Remote Storage."

- Highlight "Baud Rate."

- Define the Baud rate by pressing the **LOWER** or **HIGHER** softkeys. The Baud rate you select must match the PC's Baud rate.

- Highlight "Handshaking."(The handshaking feature verifies that **MILL*PWR*** and the PC are communicating properly during file transfers.)

- Press the **ENABLED** softkey.

- Press the **USE** key.

- Continue with Setup or press the **USE NEW SETTINGS** softkey.

Spindle Control
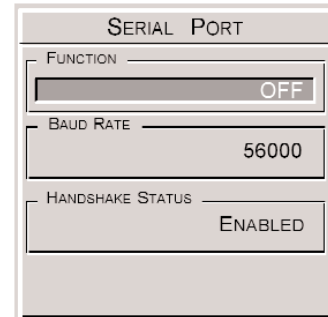
If you are using a milling machine that has an electronic variable-speed spindle, **MILL*PWR***
will soon be able to control the spindle for you.

With the spindle control enabled, **MILL*PWR*** will apply the direction and speed parameters
you enter into the "Set Tool" step of a program. Depending on the spindle's motor, you may
be able to control the spindle's speed and direction, or just the direction.

If the spindle control is *disabled*, the settings you program in "Set Tool" will be displayed
in an Operator Intervention Message (OIM) when you run the program. You will be prompt-
ed to set the speed and direction manually.

Error Log

MILL*PWR* includes a built-in error log that will record system assertion failures. If an assertion
failure is detected, **MILL*PWR*** will automatically record the date, along with a brief description
of the failure.

Messages can be cleared by pressing the **CLEAR** key, then pressing **YES** to proceed or **NO** to
cancel. It's best to let messages accumulate to establish a history.

To save these records for future reference, place a floppy disk into the disk drive and press the
SAVE FLOPPY softkey.

Servo Tuning

The servo motors for each **MILL*PWR*** system are properly tuned upon installation. However, you can fine tune your system periodically using the Auto Tuning feature. Before beginning the Auto Tuning procedure, first make sure your **MILL*PWR*** system software is version 2.2.8 or higher.

The Auto Tuning method is done completely through the **MILL*PWR*** console. Therefore, there is no need to open the servo motors. Since all adjustments are made from the console, the changes will not affect the original amplifier setting values.

When using the Auto Tuning feature, you should tune each axis (X and Y) individually—to achieve optimal performance. Both the Balance and the Gain for the axis will need to be adjusted.
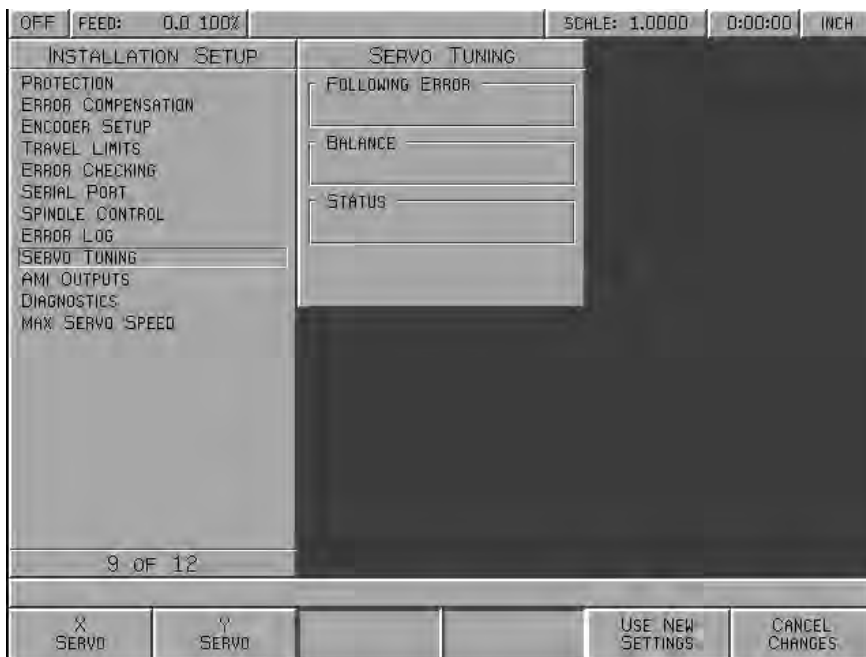
If the servo motors cannot be tuned automatically, the following message will appear: "Not enough adjustment, please use manual mode."

Please contact your ACU-RITE distributor.

Automatically Tuning the Servo in Console Mode

1. Make sure the axis you are tuning is roughly in the middle of its travel (They will be moving approximately two (2) inches in each direction.)

   Note to Installer: No RS-232 cable should not be connected to any axis. Setting value tables could become contaminated.

2. Power up the **MILL*PWR*** console, make sure the E-Stop button is out, and home the axes as you would normally.

3. Press the "Setup" key.

4. Arrow down to highlight "Installation Setup" and press **ENTER**.

5. When prompted type in the required passcode and press **ENTER**.

6. Arrow down to "Servo Tuning" and press **ENTER**.

7. Select the axis you want to tune by pressing the appropriate soft key.

8. The prompt bar should indicate you are in Console Mode.

9. Press the **AUTO TUNING** soft key, then press the **BALANCE TEST** soft key.
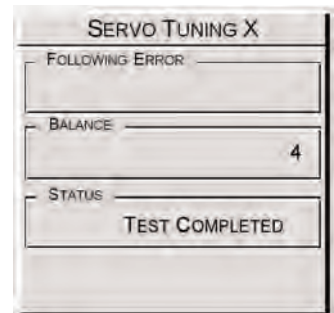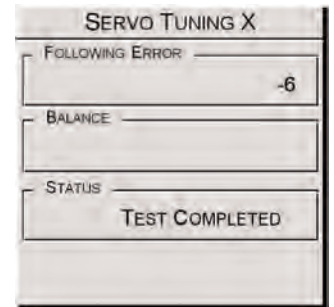
TABLE WILL MOVE AT LEAST 2" DURING THE TEST					
AUTO TUNING		STOP TEST		GAIN TEST	BALANCE TEST

10. The console will begin adjusting the balance values until it finds the value that keeps the axis as motionless as possible.

   ⚠ **Do not touch the table during the balance test.**

11. When the Auto Tuning process is completed, the prompt bar will read "Test Completed." Now proceed with the Gain Test.

12. Make sure the axis you are tuning is roughly in the middle of its travel. Press the **GAIN TEST** soft key.

13. The control will cause the axis to begin moving back and forth approximately two (2) inches in each direction.

14. While the axis is moving, the control automatically adjusts the Gain value until the "Following Error" is at its lowest value.

15. Once the control has reach the optimal Gain value, it will automatically stop axis motion and indicate it is done by displaying "Test Completed."

16. The correct values for the Gain and Balance tests have now been established. Now you will need to exit and save the new values in to the console.

17. Press the **STOP TEST** soft key, then press the **USE NEW SETTINGS** soft key.

18. Power down the console and restart to begin normal operation and have the updated values implemented.

AMI Outputs

In Position Relay

The Auxiliary Machine Interface (AMI) has the ability to send a signal to external devices. This signal is sent when **MILL*PWR*** is in position for Position Drill steps and Hole pattern steps. This is when the table has moved to either the X, Y coordinates of the position drill step or to each hole of a hole pattern.

Choose the appropriate signal for your application from the three options explained below.

ON: When the X, Y coordinates for the position drill step or when each hole of a hole pattern is reached, a signal is sent for output. This signal will continue to be output until the **GO** button is pressed, a GO signal is received through the AMI, the table is activated, or the program is stopped.

OFF: No signal is sent.

Pulsed: With this option the signals will only be sent for the timed intervals that you establish. Enter a time interval between 0.1 and 999.9 seconds. When the X, Y coordinates for the position drill step, or when each hole of a hole pattern is reached, a pulsed signal will be sent. This signal will continue until either the pulse time interval expires, the **GO** button is pressed, a GO signal is received through the AMI, the table stop button is activated, an E-Stop input is detected, the program is stopped or the program ends.
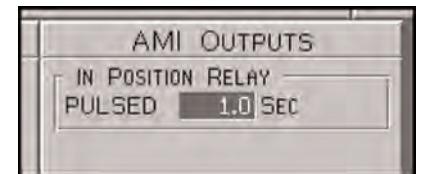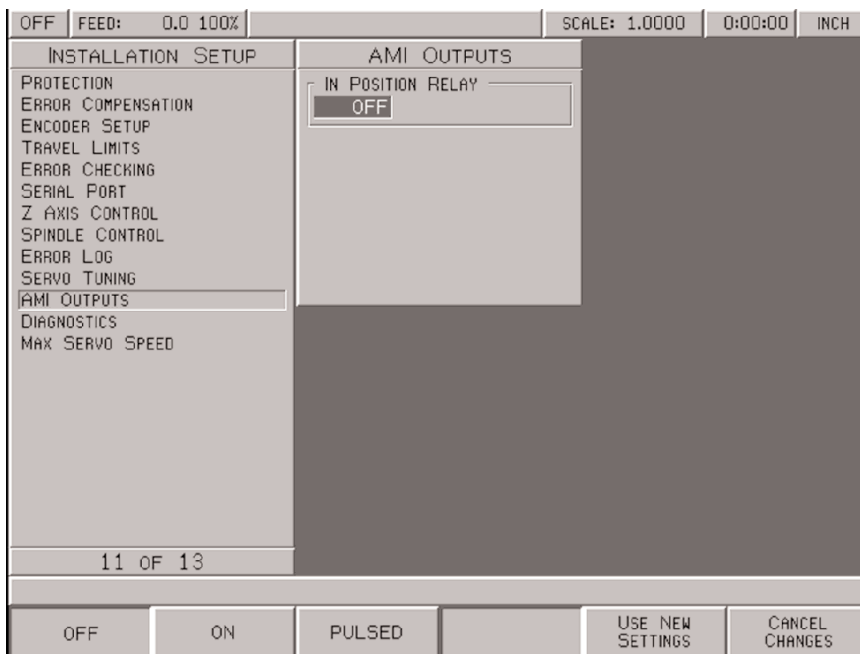
Diagnostics

The diagnostics section houses information about this system's mechanical operations, memory availability, and machine usage. Tests are also available to help you monitor the system's overall performance.
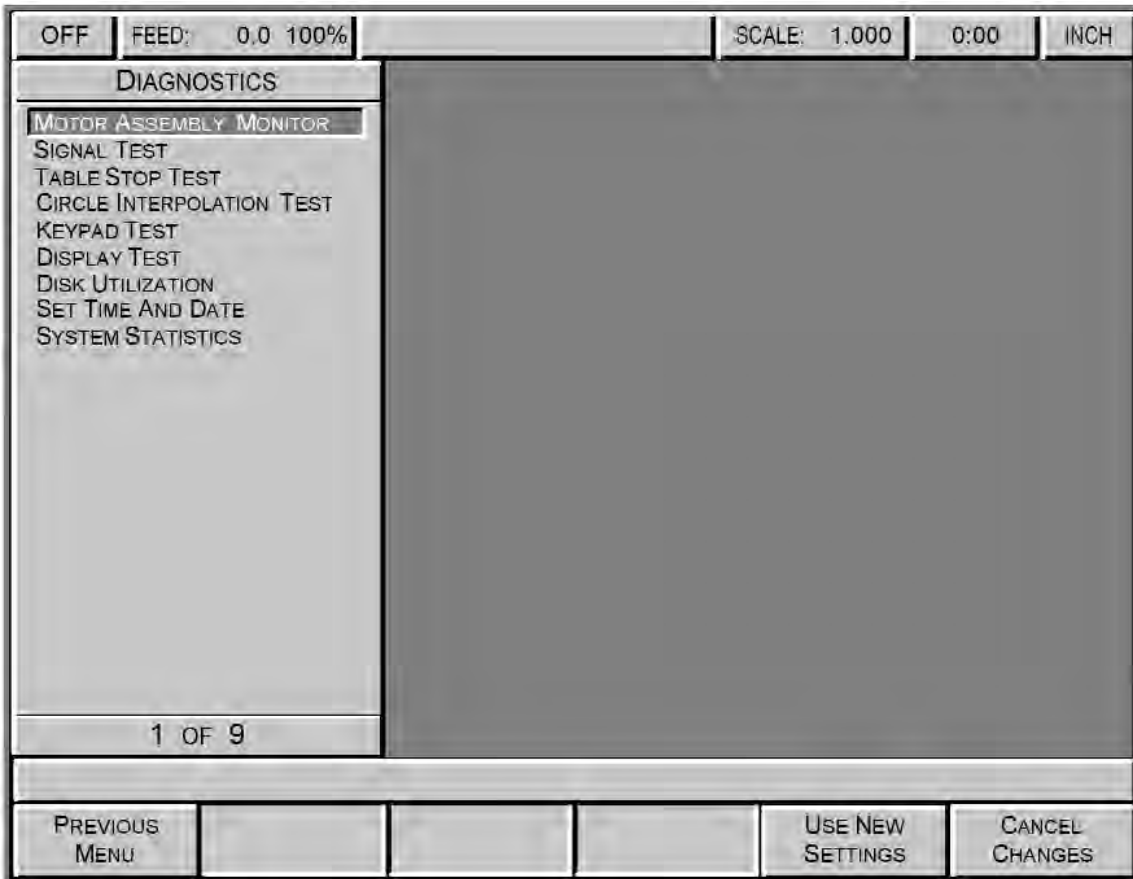
Motor Assembly Monitor

When the motor assembly monitor is enabled, it shows the real-time operating status of each axis on the DRO screen.

MOTOR ASSEMBLY MONITOR		
MONITOR		
		ENABLED

X	POWER ON	Y	POWER ON	Z	
INC	0.0000	INC	0.0000	INC	0.0000
ABS	0.0000	ABS	0.0000	ABS	0.0000
STATUS	OFF	STATUS	OFF		
TEMP	15°C	TEMP	14°C		
CURRENT	-24 0	CURRENT	17 0		

Top Bar (Power Status)	Reports on the AC power status (On or Off); also reports system failures and emergency-stop conditions when they occur.
INC	Displays the table's incremental position (its position relative to incremental zero).
ABS	Displays the table's absolute position (its position relative to datum).
Status	Reports on each motor's operating status (On, Off, Not Present or Fault).
Temp	Shows the internal air temperature for each motor. Temperatures should not exceed 75° C.
Current	Monitors the motor's current voltage. An "X" after the value indicates that the voltage inrush bypass circuit is active. It can also be seen when the E-stop button is pressed and then released. The "X" should disappear within 2 - 3 seconds after the system is powered up.

To enable or disable the motor assembly monitor:

- From the Diagnostics screen, highlight "Motor Assembly Monitor" and press **ENTER**.

- Press either the **ENABLED** or **DISABLED** softkey. (If the **ENABLED** softkey is selected, the motor assembly monitor display will appear on the DRO screen.)

- Press the **USE** key.

- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

Signal Test

The signal test shows when an axis was last tuned and whether or not the digital-to-analog converter is emitting an analog signal. When the test is complete, either "Pass" or "Fail" will appear in the "Signal Test Result" field. If a failure is reported, check the message bar prompt for details.

To display the signal test:

- From the Diagnostics screen, highlight "Signal Test" and press **ENTER**.

- Press the softkey for the appropriate axis.

- Press the **EXIT TEST** softkey to close the "Signal Test" screen.

- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey

Table Stop Test

The table stop test shows whether or not an emergency stop (E-stop) condition has been detected or the internal switch has been pressed.

E-stop Signal Status	Reports on whether or not the system detects an emergency stop condition. An E-stop condition can originate from the operator console, Z-axis, or optional AMI.
Table Stop Button	Displays the status of the table stop button (Pressed or Released) on the operator console.

To display the table stop test:

- From the Diagnostics screen, highlight "Table Stop Test" and press **ENTER**. The test will run automatically.

- When the test is complete, press the **EXIT TEST** softkey to close the "Table Stop Test" screen.

- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

```
TABLE STOP TEST
E-STOP SIGNAL STATUS
                        NO E-STOP
TABLE STOP BUTTON
                         RELEASED
Z LIMIT
                         NO FAULT
```
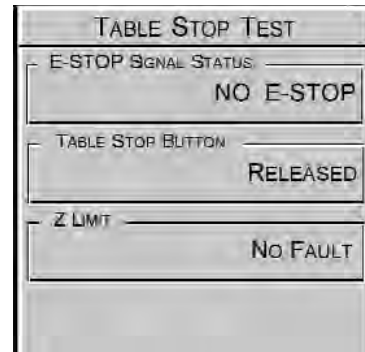
Circle Interpolation Test

MILL*PWR* includes two circle interpolation tests. The small circle test checks for machine-related problems—an unbalanced ballscrew or poor gib adjustment, for example. The large circle test checks whether or not the motors are in tune.

The circle should maintain its original shape (fluctuations up to 0.0005" are normal). If fluctuations exceed 0.0005", the system may require adjustment (refer to the **Trouble-shooting Guide** for recommended solutions). The examples on the next page illustrate some of the problems that can be detected. For accurate results, do not run the circle test while the spindle is under a cutting load.

You have the option of saving the test results onto a floppy disk for later review.

To run the circle interpolation test:

- From the Diagnostics screen, highlight "Circle Interpolation Test" and press **ENTER**.

- Check that the table has at least 3" of clearance for each axes.

- Press either the **SMALL CIRCLE TEST** or **LARGE CIRCLE TEST** softkey.

- (Optional) When the test is completed, insert a floppy disk into the **MILL*PWR*** disk drive and press the **SAVE TO FLOPPY** softkey. After **MILL*PWR*** has finished saving, remove the disk.

- Press the **EXIT TEST** softkey to close the "Circle Inter-polation Test" screen.

- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.
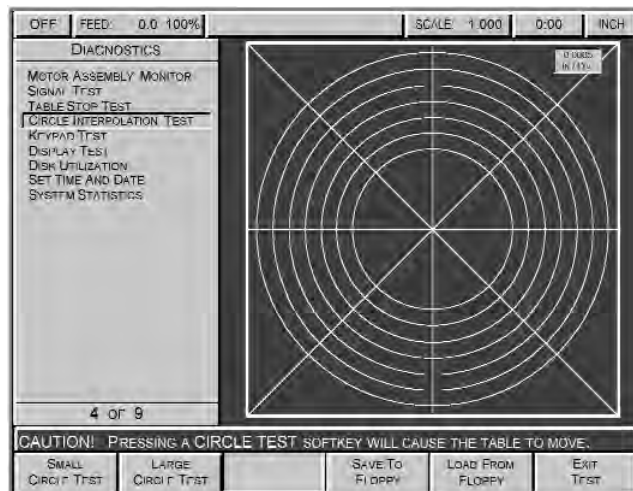
Examples of Circle Interpolation Test Results

Please note that the graphics are exaggerated for clarity.
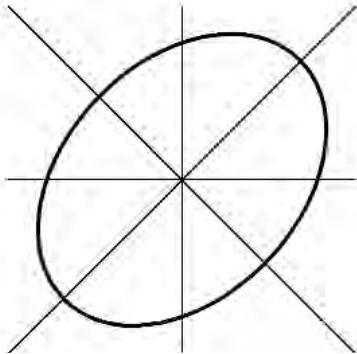
Following Error caused by servo
gain misadjustment
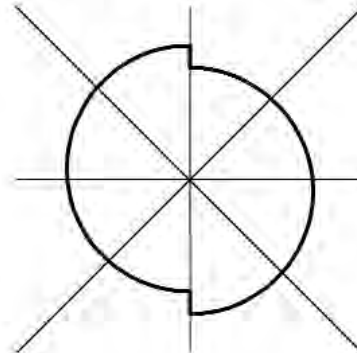
Backlash from lost motion or play
in the guideways
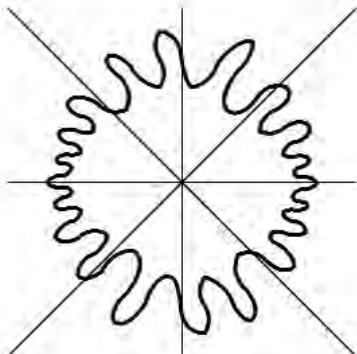
Cyclic Error caused by a flaw in the
ballscrew or ballscrew mounting
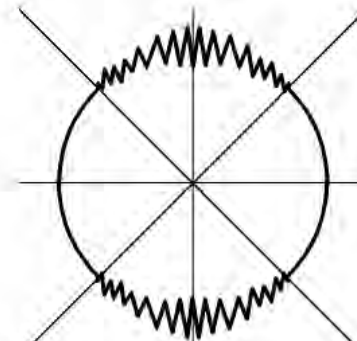
Stick-Slip of an axis, esp. at low
speeds (in this case, Y-axis)
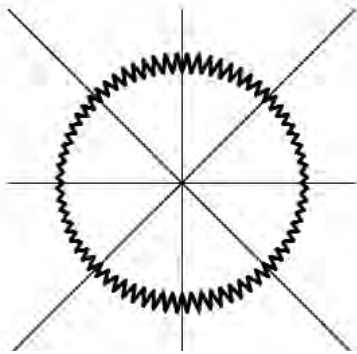
Machine Vibration in an axis
(in this case, Y-axis)
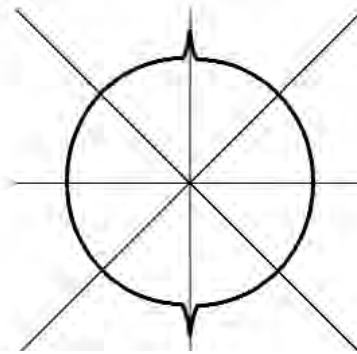
Axis Reversal Spike shows an axis
sticking when velocity reaches zero
(in this case, Y-axis)

Keypad Test

The keypad test verifies that all of the keys, the remote pendant switch and the optional electronic edge finder are functioning properly.

When you press a key, the corresponding graphic should shrink; when you release the key, the graphic should return to its full size. If the graphics don't change at all, it is likely that the key is not functioning properly. If the graphic shrinks, but does not return to normal when the key is released, the key may be sticking. Bullets (•) mark the keys that have already been tested.

To run the keypad test:

- From the Diagnostics screen, highlight "Keypad Test" and press **ENTER**.

- Press the key(s) on the keypad that you want to test. Press the remote pendant or use the electronic edge finder to test their status.

- Press the **CANCEL** key on the keypad *twice* to exit the "Keypad Test" screen.

- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.
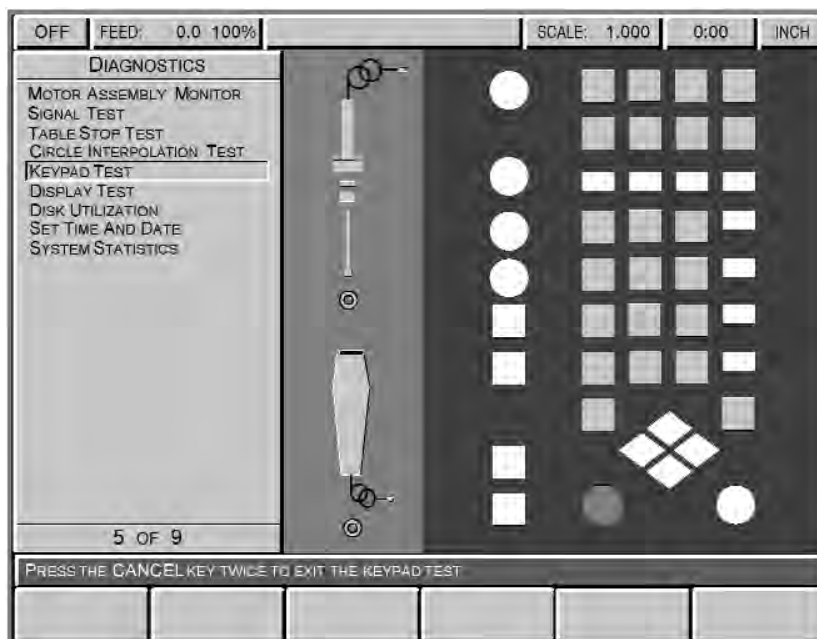
Display Test

The display test is used to verify that the system's "sleep" mode is working properly and that all of the pixels in the LCD are functioning.

Pressing the "Sleep Test" softkey will activate the sleep mode (the screen will go blank). Press any key or move the table to return the screen to its normal view.

Each time you press the "Color Test" softkey, the display will flash a different color. Keep pressing the softkey until the sequence is complete and the screen returns to the normal view.

SLEEP TEST		COLOR TEST			EXIT TEST

To run the display test:

- From the Diagnostics screen, highlight "Display Test" and press **ENTER**.

- Press either the **SLEEP TEST** or **COLOR TEST** softkey.

- If you pressed the **SLEEP TEST** softkey, press any key or move the table to return the screen to its normal view.

  If you selected the **COLOR TEST** softkey, press it repeatedly and scroll through the color sequence (black, white, red, green, blue). The test is complete when the normal view returns.

- Press the **EXIT TEST** softkey to close the "Display Test" screen.

- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

Disk Utilization

The disk utilization screen shows **MILL*PWR***'s memory capacity and how much memory is still available. Like a PC, **MILL*PWR*** uses memory each time a file is saved onto the system's internal memory. When a file is deleted, memory is restored.

Total Clusters   Total number of clusters available on the **MILL*PWR*** system. One cluster equals 8,192 bytes of memory.

Available Clusters   Number of clusters still available for use.

Total Bytes   Total amount of memory (measured in bytes) shipped with the **MILL*PWR*** system.

Available Bytes   Amount of memory (measured in bytes) still available for use.

To view the disk utilization screen:

- From the Diagnostics screen, highlight "Disk Utilization" and press **ENTER**.

- Press the **EXIT TEST** softkey to close the "Disk Utilization" screen.

- Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

DISK UTILIZATION

TOTAL CLUSTERS
41954

AVAILABLE CLUSTERS
10073

TOTAL BYTES
29888512

TOTAL BYTES AVAILABLE
20629504
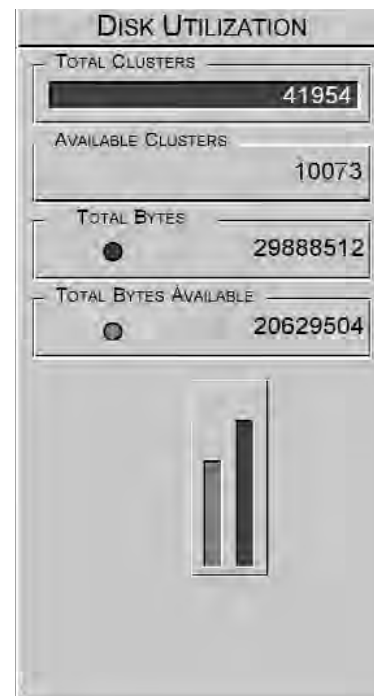
Set Time and Date

The time and date appear on the error log and on any saved part program. You can reset the time and date at any time. Enter four digits for the year.

To change a setting:

• From the Diagnostics screen, highlight "Set Time and Date" and press **ENTER**.

• Highlight the field you wish to update.

Using the keypad, enter the appropriate number.

• Press the **USE NEW SETTINGS** softkey to close the "Set Time and Date" screen.

• Continue with Diagnostics or press the **USE NEW SETTINGS** softkey again.

System Statistics

The system statistics keep track of the machine's usage. The time that the **MILLPWR** system, the display and the servo motors have been running is displayed in hours. The "distance" values for X, Y, Z and W reflect how far the scales have traveled for each axis.

To view the system statistics:

• From the Diagnostics screen, highlight "System Statistics" and press **ENTER**.

• Press the **EXIT TEST** softkey to close the "System Statistics" screen.

• Continue with Diagnostics or press the **USE NEW SETTINGS** softkey.

Max Servo Speed

```
OFF │ FEED:    0.0 100% │ TOOL:   0.0890 INCH CARB ML │ SCALE: 1.0000 │ 0:00:00 │ INCH

   INSTALLATION SETUP          MAX SERVO SPEED
 PROTECTION                  ┌ X-Y SPEED IPM ─────────
 ERROR COMPENSATION          │ ████████████████   100│
 ENCODER SETUP
 TRAVEL LIMITS
 ERROR CHECKING
 SERIAL PORT
 SPINDLE CONTROL
 ERROR LOG
 SERVO TUNING
 AMI OUTPUTS
 DIAGNOSTICS
 MAX SERVO SPEED


         12 OF 12


                                              USE NEW    CANCEL
                                              SETTINGS   CHANGES
```

MILL*PWR* can move each axis a maximum of 100 inches per minute (IPM). This parameter allows you to establish a lower maximum speed for the X, Y, axes.

To lower the maximum servo speed

- Using the keypad, enter the new speed for the X, and Y, axes in inches per minute.
- Continue with setup or press the USE NEW SETTINGS softkey.

Remote Storage

When you create programs using **MILL**PWR, you can save them on a PC using the remote storage feature. The remote storage feature enables you to free up **MILL**PWR's internal memory. Plus, a PC is the ideal place to back up important programs and organize them into one convenient location.

Equipment

To set up remote storage, you will need the following items:

- **MILL** *PWR*
- IBM-compatible PC
- Serial cable
- Remote Storage software

Choosing a Serial Cable

Before **MILL**PWR and a PC can communicate, they need to be connected to each other with a serial cable. The graphic below illustrates the necessary cable wire configuration. Check that the cable you purchase conforms to this wiring diagram:
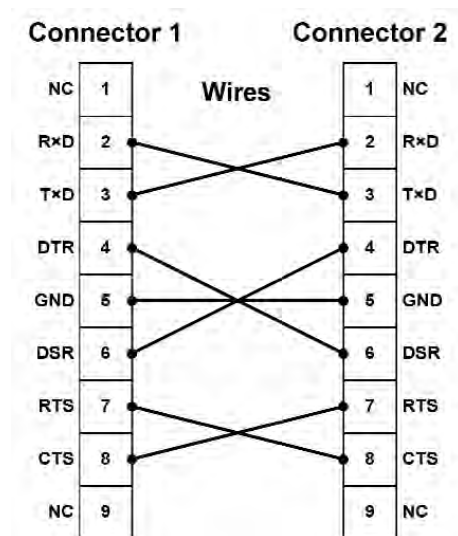
Connecting MILL*PWR* to a PC

> ⚠️   Power down both **MILL*PWR*** and your PC before connecting
> the serial cable.

Connect one end of the cable into the serial connector located on the rear of **MILL*PWR***
operator console. Connect the opposite end of the cable into an available COM port on
a PC.

Note: The COM port must be *available*—that is, not already assigned a piece of hardware.
You cannot simply disconnect a piece of hardware and connect the serial cable. If
there's no COM port recognized as being available by a PC, then you will need to
make one available (consult a PC owner's manual).
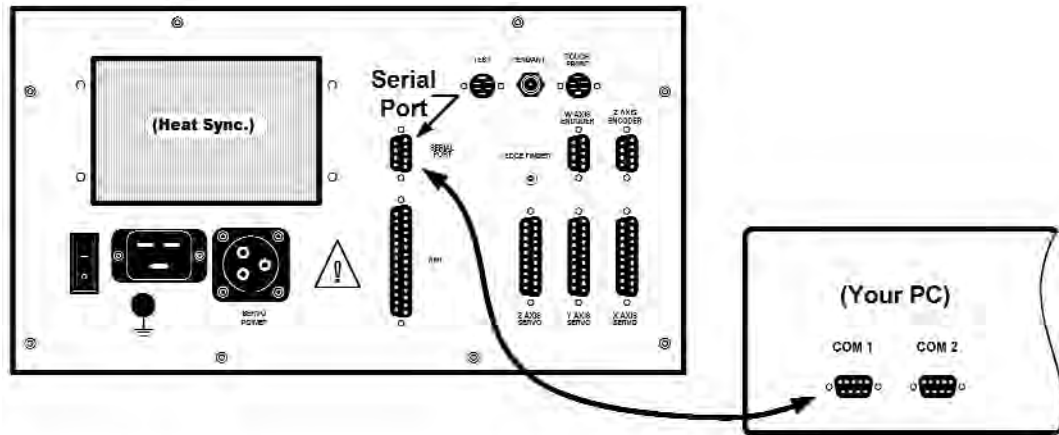
Installing the Remote Storage Program onto a PC

You can install the Remote Storage program onto a PC using either Microsoft®
Windows® 95/98 or MS DOS® operating systems. Choose the system you will use, then
follow the applicable steps below.

For Windows 95/98:

1. Power up the PC.
2. Insert the "Remote Storage" disk into the PC's floppy disk drive.
3. Click on "START."
4. Click on "Run," then type **a:install** and click "OK." This copies all of the files from the disk and onto your PC.
5. Instructions on how to use Remote Storage will be displayed on the PC screen.
6. Close the DOS window. A Remote Storage icon should now appear on the PC screen. Double-click on the icon to run the Remote Storage program.

For MS DOS:

1. Power up your PC.
2. Insert the "Remote Storage" installation disk into the PC's floppy disk drive.
3. At the C:\. prompt, type **a:** then press **ENTER**.
4. After the A:> prompt, type **install**, and press **ENTER** again. This command tells the PC to copy all the files from the disk onto your PC.
5. At the next prompt, type **c:** then press **ENTER**.
6. After the C:\ prompt, type **cd remtstor** and press **ENTER**.
7. C:\REMTSTOR\> will appear. Type **remtstor** and press **ENTER**.

The PC should now display the Remote Storage Utility screen.

Setting up a COM Port and BAUD Rates

After the Remote Storage software has been installed on a PC, the following screen will appear:
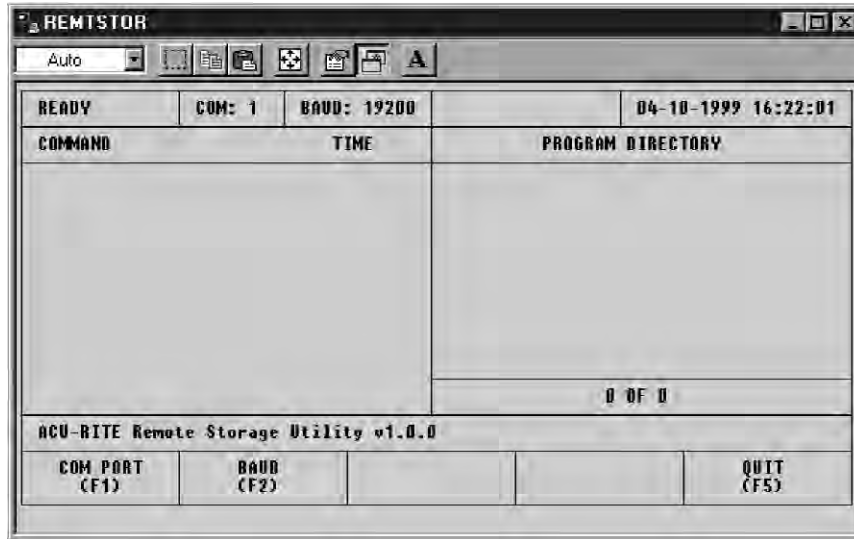
Now you need to identify a COM (short for "communication") port and set the BAUD rate.

- **COM Port** - COM port 1 is the program's default setting. You can switch to COM port 2 by pressing the F1 key on the PC's keyboard. Remember that the COM port setting must be available, and it must correspond with the PC port that you connected the serial cable to.

- **BAUD Rate** - Set the BAUD rate by pressing the F2 key on your PC's keyboard and scrolling through the list of BAUD rates available until the correct value appears. Check that the BAUD rate you define for your PC matches the BAUD rate you set for **MILL*PWR***.

After the COM port and BAUD rate have been set, you can begin storing part programs onto your PC using **MILL*PWR***'s serial port. (For more information about creating, saving, and deleting programs, refer to **Programming**.)

Activating the Remote Storage Feature in MILL*PWR*

Refer to **Setup** to activate the remote storage feature in the **MILL*PWR*** system.

Troubleshooting

If the communication link between **MILL*PWR*** and the PC is disrupted, an error message will appear on the **MILL*PWR*** screen. Settings that do not match and an improper serial cable connection are the most common problems and are the easiest to correct. Refer to the general **Troubleshooting Guide** section for details.

This page intentionally left blank.

Troubleshooting Guide

Introduction

This troubleshooting guide is intended to assist you with diagnosing problems should you experience any difficulties with the **MILL*PWR*** system.

Using the Table

This troubleshooting guide is arranged in three columns entitled Symptom, Probable Cause and Recommended Corrective Action. The Symptoms are listed in the order of most common, easiest to check, and least expensive to correct. First, locate the symptom that best describes the problem that needs to be corrected. Then, identify the probable cause that most closely matches the problem that you are experiencing and implement the recommended corrective action.

This troubleshooting guide pertains to the ACU-RITE **MILL*PWR*** system specifically and on occasion the machine tool itself. It does not fully address problems associated with inadequate tooling, improper feed rates and/or spindle speeds, etc.

Note: This troubleshooting guide has been written with the understanding that your ACU-RITE **MILL*PWR*** system has been properly installed by a factory-trained and certified technician.

IMPORTANT
Do not attempt to make any repairs to the operator console and/or motor assemblies on your own. Please contact your ACU-RITE distributor for repair and/or replacement assistance.

TROUBLESHOOTING GUIDE

Symptom	Probable Cause	Recommended Corrective Action
Operator console displays a blank screen.	System has been idle for approximately 20 minutes and is in screen saver mode.	Check that the power indicator light, located in the upper left-hand corner of the operator console, is illuminated. If the power indicator light is illuminated, either move the table or press any key on the operator console to restore the display. If the power indicator light is not illuminated, power has been interrupted. Follow the recommendations for "No Power."
	No power	Check that the power switch located on the rear of the operator console is on. Check that the power cord is properly connected. Check that there are no blown fuses or tripped breakers. Check that the main power source meets the specifications required of the system. In the event of a power interruption, you simply have to wait until the power is restored.

Symptom	Probable Cause	Recommended Corrective Action
Operator console displays a blank screen (*continued*)	LCD display failure	Contact your ACU-RITE distributor for repair and/or replacement procedures
Operator console displays a flashing cursor	A floppy disk is in the floppy disk drive	Remove the floppy disk from the disk drive and cycle power to the system. Remember to use the "Find Home" routine.
	System begins to power up, but cannot get past flashing cursor	The system software has been corrupted or may not have been properly installed. Re-install the system software and power up the system again. Remember to use the "Find Home" routine.
	The floppy disk drive is not functioning properly	Power up the system again and check that the LED (located on the front of the floppy disk drive) is illuminated. Also, listen for the spinning sound that a floppy disk drive makes when it is being accessed. Should the above recommendations not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.

Symptom	Probable Cause	Recommended Corrective Action
Operator console keypad does not function properly.	System needs to be reset.	Power down the system. Wait for approximately one minute then power the system up. Remember to reestablish machine tool zero using the "Find Home" routine.
		Run the Keypad Test (refer to **Diagnostics**, Section 7).
		Should the above recommendation not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
System powers up, but motor(s) will not move table	Emergency table stop button has been pressed.	Check the message bar for the following message: "TABLE STOP BUTTON IS PRESSED."
		Release the emergency table stop button and press the **CLEAR** key.
	GRAPHICS ONLY selected in "Run Options."	Review selections in "Run Options."
	MANUAL POSITIONING selected in "Run Options."	Review selections in "Run Options."
	Improper scale connection	Check the message bar for the following message: "DSP fault-(X or Y) axis position error check failed."
		Check the scale connections at the motor assemblies.

Symptom	Probable Cause	Recommended Corrective Action
System powers up, but motor(s) will not move table (*continued*)	X- and/or Y-axes precision glass scales stopped counting	While in the DRO screen, move each axis manually to check if the precision glass scales count. If the precision glass scales do not count, perform the following:
		• Press the emergency table stop button.
		• Swap the X- and Y-axes servo connections.
		Do not release the table stop button until you have restored the X- and Y- servo connectors to their proper locations!
		• In the DRO screen, move each axis manually to check if the precision glass scales count. If a precision glass scale doesn't count, follow the instructions for "The scale(s) miscount in certain areas only" below.
	The scale(s) miscount in certain areas only	• Remove the scale(s). Clean the glass (located inside the aluminum extrusion) and the reading head glass using a cotton swab and denatured alcohol or acetone.
		• Re-install the scale(s) and realign the reading head(s).

Symptom	Probable Cause	Recommended Corrective Action
System powers up but motor(s) will not move table (*continued*)	The scale(s) miscount in certain areas only (continued)	Should the above recommendation not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
	Improper X- and/or Y-axes servo connection.	Message bar reads: "(X OR Y) SERVO UNPLUGGED FAULT." Check the X- and Y-axes servo connections located on the rear of the Operator Console.
	Motors are not receiving power	Check the power connection at each motor and at the Servo Power connector on the rear of the Operator Console. Enable the Motor Monitor (refer to **Diagnostics**, Section 7).
Table run away	The X- and Y-axes servo connections located on the rear of the Operator Console are reversed.	Check that the cables are properly connected to the correct axes.
Table will move in only one direction.	Table is beyond table travel limits	Move the table within the table travel limits. You may want to consider adjusting your table travel limits. If so, refer to **Setup**).

Symptom	Probable Cause	Recommended Corrective Action
Table has stiff movement.	Table is locked.	Check the table locks and loosen as needed.
	Gibs are worn or are out of adjustment	Refer to the machine tool operation manual for adjustment procedures.
	Not enough oil in oil reservoir or oil lines are blocked or disconnected.	Check that there is sufficient oil in the oil reservoir.
		Should the above recommendation not correct the problem you are experiencing, contact your ACU-RITE distributor for repair and/or replacement procedures.
System not repeating or circles are not round.	Machine tool related problems	Check that the knee, ram and head bolts, turret to column bolts and head to knuckle bolts are properly fastened and secure.
		Check the gibs for wear and for proper adjustment. Refer to the machine tool operation manual.
		Tram the head. Lower the quill full travel with a sturdy tool inserted into the spindle. Set up a dial indicator to check movement of the spindle, then move the tool side-to-side. Total movement should be within the machine tool manufacturer's specifications between the spindle and the spindle bearings and between the quill and the bore. You may need to replace the spindle bearings.
		Run the Circle Interpolation Test (refer to **Diagnostics**, Section 7).

Symptom	Probable Cause	Recommended Corrective Action
System not repeating or circles are not round. (*continued*)	Machine tool related problems (*continued*)	Machine tool wear. **MILL*PWR*** can compensate for machine tool wear through its linear error compensation (LEC) feature. (refer to Setup).
	Position feedback from precision glass scales	Check each precision glass scale and reading head for proper installation. Check that the mounting brackets and fasteners are tight and secure.
	System is out of tune.	Perform the auto tuning procedures for both the X- and Y-axes.
		If auto tuning the system does not correct the problem, contact your ACU-RITE distributor.
Incorrect dimensions	Programming error	Check that the proper dimensions were entered for each step of the program.
		Check that the correct tool information was entered for each step of the program.
		Measure the tool diameter to check for tool wear.
		Check that the correct scale factor is being used.

Symptom	Probable Cause	Recommended Corrective Action
Poor finish	Dull tool	Replace or sharpen the tool.
	Incorrect feed rates and/or spindle speeds	Check that the correct feed rates have been entered for each step of the program. Check the spindle speed setting.
	Gibs are worn or are out of adjustment.	Refer to the machine tool operation manual.
	Worn spindle bearings	Lower the quill full travel with a sturdy tool inserted into the spindle. Set up a dial indicator to check movement of the spindle; then move the tool side-to-side. Total movement should be within the machine tool manufacturer's specifications between the spindle and the spindle bearings; and between the quill and the bore. You may have to replace the spindle bearings.
	System is out of tune.	Perform the auto tuning procedures for both the X- and Y-axes.
		If auto tuning the system does not correct the problem, contact your ACU-RITE distributor.

Symptom	Probable Cause	Recommended Corrective Action
BAUD rate error	**MILL*PWR*** and PC BAUD rate settings do not match.	Change one of the settings so that the BAUD rates for **MILL*PWR*** and the PC are the same.
Check sum error	**MILL*PWR*** and PC BAUD rate settings do not match.	Change one of the settings so that the BAUD rates for **MILL*PWR*** and the PC are the same.
No response from PC	BAUD rate is too slow or too fast.	Adjust the BAUD rates on both **MILL*PWR*** and the PC. Check that they match when you are finished adjusting them.
	MILL*PWR* and PC BAUD rate settings do not match	Change one of the settings so that the BAUD rates for **MILL*PWR*** and the PC are the same.
	Incorrect serial cable	Verify that you are using the correct type of serial cable. Refer to page 8-1, choosing a serial cable.

| | | Recommended |
Symptom	Probable Cause	Corrective Action
No response from PC (cont'd)	The PC's COM port setting is different from the COM port that the serial cable is connected to.	If the COM port setting (1 or 2) on the PC screen does not correlate with the COM port that the serial cable is connected to:

Change the PC's COM port setting;

 -OR-

Disconnect the cable from the COM port that it is connected to and connect it to a different COM port. |
| | The selected COM port is disabled. | Change the PC's COM port setting and connect the serial cable into the appropriate port;

 -OR-

Exit the remote storage program on the PC, then refer to the PC's operation manual for instructions on how to enable a COM port. |
| Time-out error | Loose serial cable | Check that the serial cable is properly connected to **MILL*PWR*** and to the PC. |

This page intentionally left blank.