EZ-PATH I

OPERATIONS

AND

PROGRAMMING MANUAL

March 1996

Bridgeport®

Code No. 11865457
Rev. B

TABLE OF CONTENTS

Chapter 5
EZ-PATH TOOL LIBRARY

Chapter 6
EZ-PATH DO EVENT MODE

Chapter 7
EZ-PATH MDI MODE

Chapter 8
EZ-PATH EDIT MODE

Chapter 9
EZ-PATH RUN MODE

Chapter 10
EZ-PATH UTILITIES

Chapter 11
EZ-PATH TEACH MODE

Chapter 12
EZ-PATH PREVIEW MODE

INTRODUCTION

The EZ-PATH I is an extension of the Bridgeport PCNC (PC-based Numeric Control) product line, providing the familiarity of a manual lathe with the power of a CNC. The EZ-PATH is specifically designed for the first-time PCNC user and is intended to bridge the gap between the hand-wheel driven lathe and the full-featured PCNC, with the ease of use of one, and the increased productivity of the other.

The EZ-PATH is targeted at cutting one-of-a-kind parts or small job lots and gives the user the flexibility of four different machines in one package:

- In the intelligent DRO (digital readout) mode, the operator can use the advanced digital readout and the axes handwheels in the same way that the conventional lathes in the shop are operated.
- In the MDI (manual data input) mode, a step-by-step conversational display prompts the operator for all the information required to easily machine complex parts, including roughing, facing, grooving and threading operations.
- In the DO EVENT mode, the operator can switch between manual and automatic operation to make the part in a way that is the most convenient for the operator. Complex operations such as turning a radius, or a taper become very simple.
- In the TEACH mode the operator can manually turn the first part and save the coordinates of each move to replay the operations for subsequent parts.

The EZ-PATH requires no prior knowledge of CNC programming, or computer experience. Following the on-screen prompts, and entering the requested information, the operator can begin cutting a part after only a few minutes of basic explanation on the machine operation. The programming environment in the EZ-PATH intelligently prompts the user for basic part information found directly on a blueprint, and even provides math help functions for calculating necessary points.

About This Manual

This manual provides the necessary information to run and program the EZ-PATH. It provides information via illustrations, and through a complete step-by-step tutorial which actually produces a part. For users with no previous experience on the EZ-PATH, it is suggested that this manual be read in the following order for maximum clarity.

1) Chapter 1 - EZ-PATH Hardware
2) Appendix D - Axes and Coordinates
3) Chapter 2 - Starting Up the EZ-PATH
4) Chapter 3 - EZ-PATH Tutorial
5) Chapter 4 - Basic Operations

CHAPTER 1
EZ-PATH HARDWARE

This section discusses the EZ-PATH hardware. For a more complete description of the EZ-PATH's features and controls, refer to the EZ-PATH Maintenance and Installation Manual.

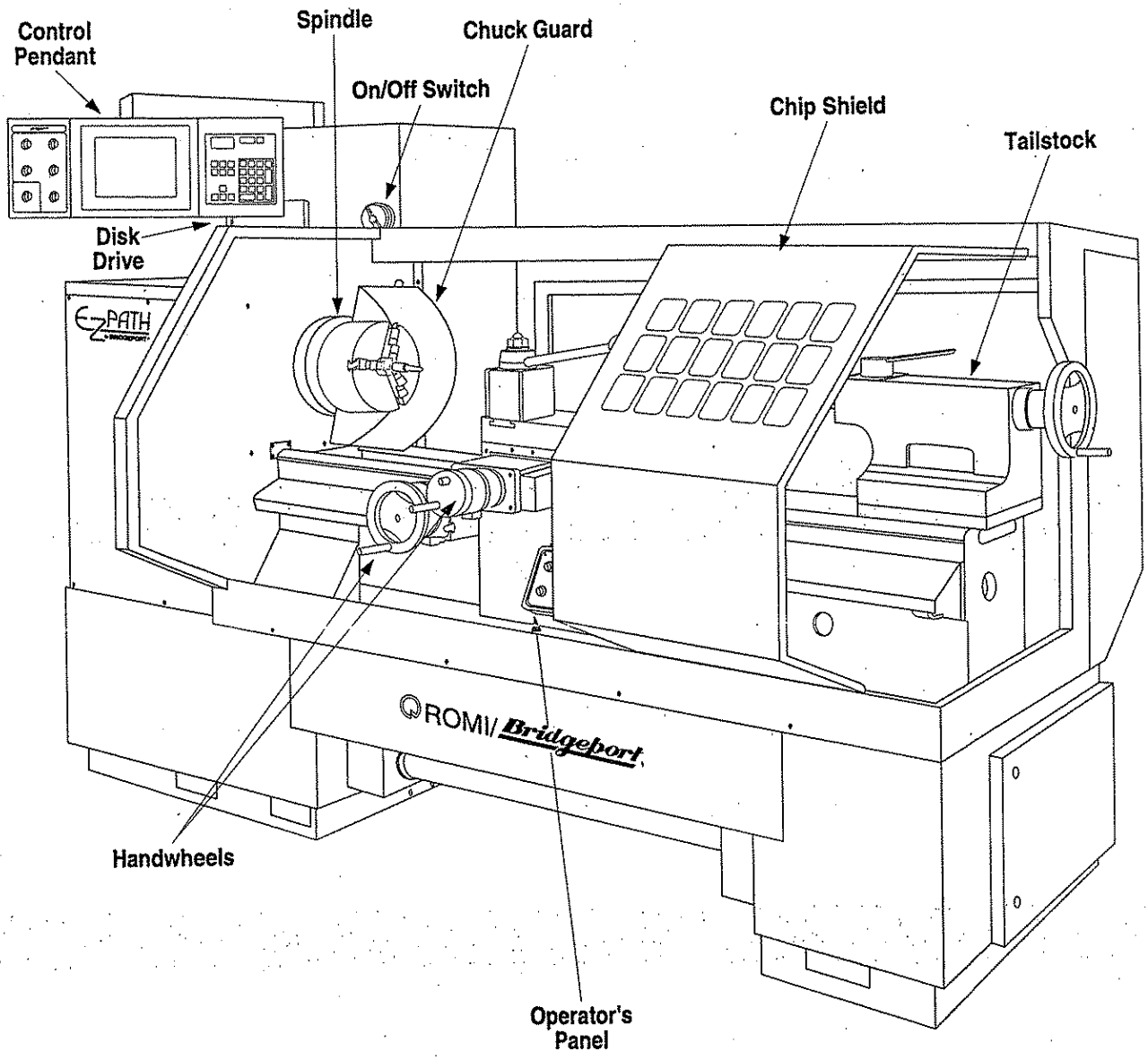


Figure 1-1

The EZ-PATH basic operation controls are described here as they are seen from the front of the machine looking from left to right.

EZ-PATH CONTROL PANEL

The EZ-PATH control panel houses the Cathode Ray Tube (CRT) display and is suspended from the pendant arm on the left side of the EZ-PATH, above the spindle housing. This panel also has six items on the left side of the CRT display, five operator control buttons and one indicator light. See Figure 1-2.

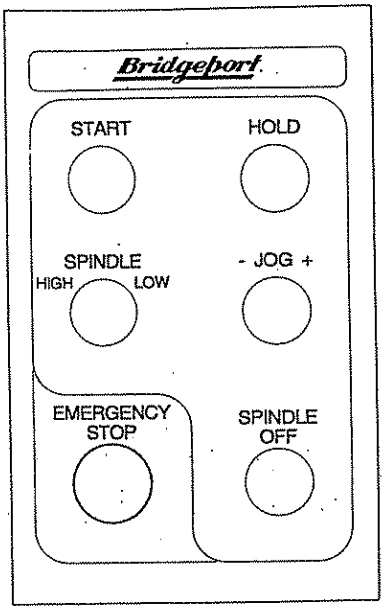


Figure 1-2

START The **START** button is used to signal the EZ-PATH that the operator is ready to begin an operation. This button is the same as pressing the **START** button on the operator's panel at the lower right side of the carriage. In the **RUN** mode, the screen prompts the operator when it expects a **START** command. Pressing this button or the + key on the operator's keyboard, in the **RUN** mode only, will cause the machine to begin the operation.

HOLD This button is a **FEED HOLD** button. This button stops the tool carriage motion wherever it is, and suspends the lathe operation. The spindle is **not** affected by the **HOLD** button.

SPINDLE OFF The **SPINDLE OFF** light is meant to show the operator that the spindle is turned off. When the lamp is lit (bright green), the power to the spindle is shut off. Only when this light is **on** is it safe for the operator to work near the spindle.

– JOG + The **JOG** button is used to move the **Z axis**. When the EZ-PATH is in **RUN** mode this key is disabled. When the EZ-PATH is in **JOG** mode, this key may be used to jog the **X axis**. The default mode of this key moves the **Z axis**. The direction of travel for the selected axis (**Z** or **X**) is shown by the arrow on the front of the button. Turn the button to the opposite sign to change the direction of travel.

EMERGENCY STOP The **EMERGENCY STOP** button is used to halt all action of the lathe in the case of an emergency. When this button is pressed, the axis motion stops, the spindle stops and the screen displays an **ALARM** message. To clear the **ALARM**, first clear the situation, (remove the part or tool, or other obstruction) and then pull the **EMERGENCY STOP** button out. The screen will prompt the operator to press a key on the keypad to confirm that the emergency has passed. Press the **ENTER** key. Operation will resume when the spindle is started and the **START** button is pressed.

SPINDLE The **SPINDLE** control has two settings, **LOW** and **HIGH**. These change the speed of the spindle according to the gearing and spindle speed charts shown on the front of the machine (Figure 1-3 and Figure 1-4). If the spindle is in neutral the machine will not perform program functions. Be sure the spindle is in gear before attempting to run a part program or use DO EVENT.

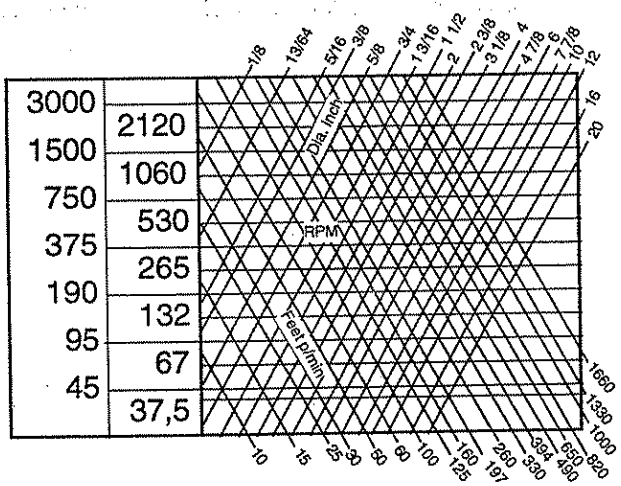


Figure 1-3

yellow / green	red / blue	I / II / III	H	L
green	blue	III	3000	1500
yellow	blue	III	2120	1060
green	blue	I	1500	750
yellow	blue	I	1060	530
green	blue	II	750	375
yellow	blue	II	530	265
green	red	III	375	190
yellow	red	III	265	132
green	red	I	190	95
yellow	red	I	132	67
green	red	II	95	45
yellow	red	II	67	37,5

Figure 1-4

HANDWHEELS

The EZ-PATH has two handwheels at the front of the machine, just below the tool carriage as shown in Figure 1-5. The handwheel to the left moves the tool carriage left and right along the **Z** axis, the upper handwheel to the right moves the tool in and out along the **X** axis. These are free to move the tool as long as no command is active, and as long as no program is running. If a DO EVENT command is active, the handwheels may move the tool in different ways. See the DO EVENT chapter in this manual for more details.

OPERATOR'S PANEL

The **OPERATOR'S PANEL** is below and to the right side of the tool carriage. There are four controls on this panel. See Figure 1-5.

SELECT The **SELECT** button is used in the execution of some **DO EVENT** commands. The **TAPER**, **CHAMFER**, and **RADIUS** commands change the way the tool is moved by the handwheels. Since the tool must also be positioned with the handwheels for these functions, the **SELECT** button is used to temporarily suspend these functions. Pressing the **SELECT** button a second time re-enables the active command. See the DO EVENT chapter in the manual for more details on these commands.

EMERGENCY STOP The **EMERGENCY STOP** button is used to halt all action of the lathe in the case of an emergency. When this button is pressed, the axis motion stops, the spindle stops and the screen displays an **ALARM** message. To clear the **ALARM**, first clear the situation, (remove the part or tool, or other obstruction) and then pull the **EMERGENCY STOP** button out. The screen will prompt the operator to press a key on the keypad to confirm that the emergency has passed. Press the **ENTER** key. Operation will resume when the spindle is started and the **START** button is pressed.

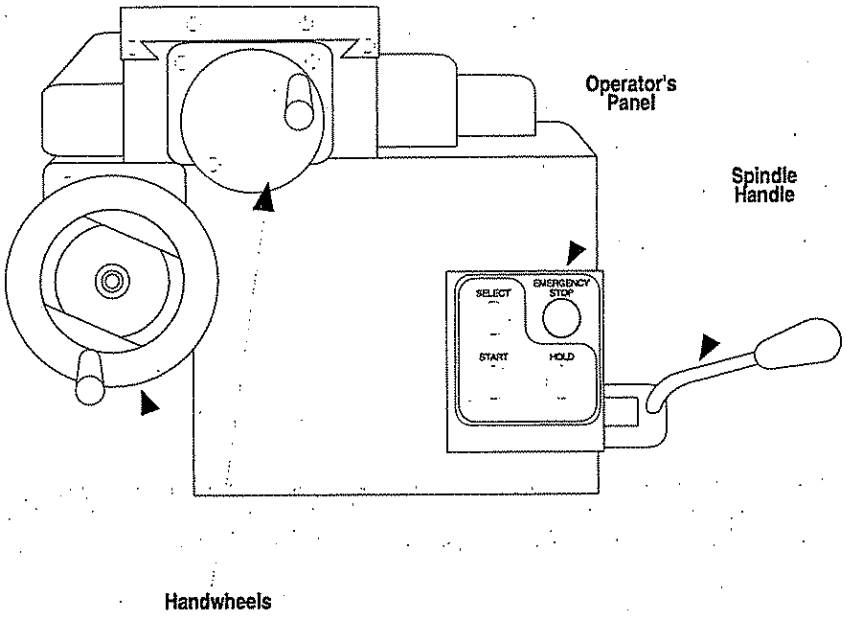


Figure 1-5

START The **START** button is used to signal the EZ-PATH that the operator is ready to begin an operation. This button is the same as pressing the **START** button on the control panel. In the **RUN** mode, the screen prompts the operator when it expects a **START** command. Pressing this button or the + key on the operator's keyboard, in the **RUN** mode only, will tell the machine to begin the operation.

HOLD This button is a **FEED HOLD** button. This button stops the tool carriage motion wherever it is, and suspends the lathe operation. The spindle is **not** affected by the **HOLD** button.

SPINDLE HANDLE

To the right side of the **OPERATOR'S PANEL** is the **SPINDLE HANDLE**. This handle is used to start, stop, and reverse the spindle. To start the spindle, move the handle to the left and up. To stop the spindle, move the handle down and to the right. This will engage the brake. The spindle is stopped immediately. To reverse the spindle motion, (turn the spindle in the opposite direction) move the handle to the left and downward (from the stop position). While the handle is in the stop position the brake is engaged. If the handle is moved either up or down without being moved left, the brake disengages but the spindle does not start.

EZ-PATH COMPUTER HARDWARE

For detailed descriptions of the EZ-PATH computer hardware, see the Appendix titled System Overview at the back of this manual. Also, refer to the *Parts Breakdown Manual* (code no. 11865458), *Maintenance Manual,* (code no. 11865459), and *Electrical Manual* (code no. 11865460) for more details.

WARNING:

The EZ-PATH screen is subject to phosphor burn-in, a condition which often occurs on CRT (Cathode Ray Tube) screens when they display the same information for long periods of time. To prevent the screen from becoming damaged in this manner it is important that you turn down the screen intensity, especially when the machine is not in use for long periods of time. To help extend the life of the computer screen, a screen saver has been installed and will turn on after approximately three (3) minutes if the machine is not used. This will extend the life of the computer screen by a great deal. The brightness intensity control is just beneath the lower edge of the computer screen, towards the right side. The screen saver will turn off if an axis handle is moved or if a key on the keypad is struck. This first keystroke will not effect or alter any program screens or operations.

KEEP THE SCREEN INTENSITY LOW

CHAPTER 2
STARTING UP THE EZ-PATH

Introduction

This chapter discusses starting up the EZ-PATH and using the basic commands to control the movement of the tool carriage. It is important to read through this chapter before cutting any part on the EZ-PATH.

Turning on the EZ-PATH

On the panel just behind the computer screen is a large handle. This is the power switch (main disconnect) for the machine. It turns on power to the computer as well as the drive motors and spindle motor.

Turn the power handle to the **on** position shown by the arrow. You will feel a solid click when the power is turned on.

As the computer starts up, you will see various messages flash across the CRT display on the operator's panel. These messages are unimportant under normal circumstances. When full power is reached, the screen shown in Figure 2-1 is displayed.

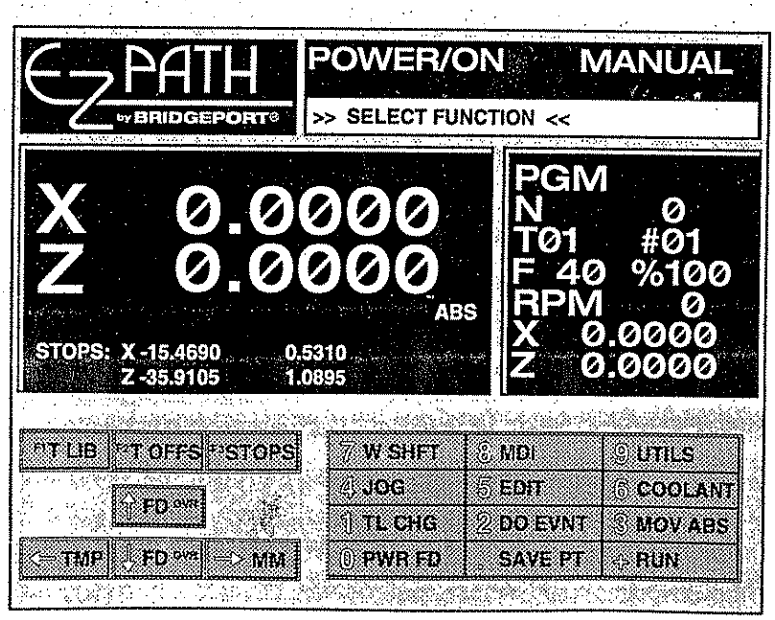


Figure 2-1

Look first at the top left of the screen and note that the message reads **EZ-PATH NOT HOMED. HIT [MOV ABS]**. This means that the EZ-PATH must go through the process of homing the axes before the machine is ready to operate. Read the section below titled **Homing the Axes** before continuing.

Homing the Axes

The EZ-PATH needs to find the limits of the carriage and cross slide travel each time the machine is turned on. This process is called **homing the axes**.

NOTE: The tailstock may interfere with the homing process. Before homing the axes, be certain that the tailstock is moved to the extreme right end of the lathe bed so that it hangs over approximately one (1) inch. In this position, it will not interfere with the homing process.

Look at the screen again. On the lower half is a pattern of buttons that represent the operator's keyboard. These buttons show what function each button has at any one moment. Right now, the **3** key on the keypad has the function **MOV ABS**. Press this key to home the axes. Press the + key to start the homing process.

The carriage immediately begins to move to the front and to the right. When the trip-switches beneath the carriage track are tripped, the carriage stops moving and the travel limits are set. This also resets the X and Z coordinates, so that the last saved origin is active. The BASIC OPERATIONS screen is displayed when homing is complete.

It is possible to by-pass the homing operation for emergency situations where power to the EZ-PATH has been interrupted while the machine was performing an operation (such as boring or drilling). *This is only used to move the tool away from the work piece in such cases where homing the machine will cause the tool to crash the work piece and cause damage to the machine.*

To by-pass the homing operation hit the **JOG** key at the **EZ-PATH NOT HOMED. HIT [MOV ABS]** message box. The screen will read: >> HIT [MOVE ABS] MOVING AXES TO HOME POSITION. Press the **ESC** key to prevent the machine from homing. It is possible that an ALARM condition will occur. To clear the alarm, press any key. Now it is possible to use either the jog knob or the hand wheels to move the tool away from the work piece. When the tool is in a safe position, press **0 EXIT** to return to the BASIC OPERATIONS screen, then hit **3 MOVE ABS** to home the machine.

WARNING!! By-passing the homing procedure does not set the machine limits. Severe damage and injury can result from using the machine without set limits. **HOME THE MACHINE BEFORE PERFORMING ANY FURTHER OPERATIONS!**

Moving the Axes

The handwheels at the front of the EZ-PATH control the movement of the tool carriage. These handwheels are **not** directly connected to the motors which move the carriage. Instead, these handwheels are connected through electronic circuits to the computer which

controls the carriage movement. This allows the EZ-PATH to change the way that the axes move in some commands. When no command is active the handwheels move the axes just as though the EZ-PATH were a manual machine. This makes the EZ-PATH very versatile as well as easy to use.

Try moving the carriage back and forth in both the X and Z directions using the handwheels at the front of the machine.

When the BASIC OPERATIONS screen is displayed the handwheels are free to move the axes as though it were a manually operated lathe. It can be used as a manual lathe, as long as no command is active.

The **JOG SWITCH** on the operator's panel next to the CRT display is also active when the BASIC OPERATIONS screen is displayed. It will move the Z axis in which ever direction is selected (+ or –). The X axis cannot be moved with the **JOG SWITCH** unless the machine is in the **JOG** mode.

Jogging the Axes

From the BASIC OPERATIONS screen, the **JOG** mode can be chosen by pressing the **4 JOG** kev. Press the **4 JOG** kev now.

To **jog** the axes means that the X and Z axes can be selected separately and can be moved by using either of two different controls on the operator's panel. The first is the **JOG SWITCH**, the second is the **STEP** + and – keys (left arrow and right arrow) on the keyboard.

The **JOG** mode defaults to the **JOG X** mode so that only the **X** axis can be jogged. Turn the jog switch so that the arrow points towards the – sign. Press the button firmly to jog the X axis. Release the button after moving the carriage several inches. Turn the switch back to the + sign, and jog the carriage back.

The **<up arrow>** and **<down arrow>** keys on the keyboard are **FEED OVERRIDE** keys. They are used to change the rate at which the carriage moves when the **JOG SWITCH** is pressed. Press the **<down arrow>** key to slow down the jog rate, then press the jog switch.

Press the **JOG Z** key (the * key) on the keyboard. Using the **jog switch**, jog the carriage several inches in the – direction. The jog direction can be changed by turning the **jog switch** towards the + sign. Jog the Z axis back towards the center.

Also try pressing the **STEP+** and **STEP–** keys. These keys jog the axes 0.0005" each time the key is pressed. Press the **0 EXIT** key to exit the **JOG** mode.

Reading the CRT Display

The EZ-PATH uses a Cathode Ray Tube (CRT) to display most of the necessary operating information. This is the screen located next to the keyboard. Learning to read this display quickly and accurately will make using the EZ-PATH much easier.

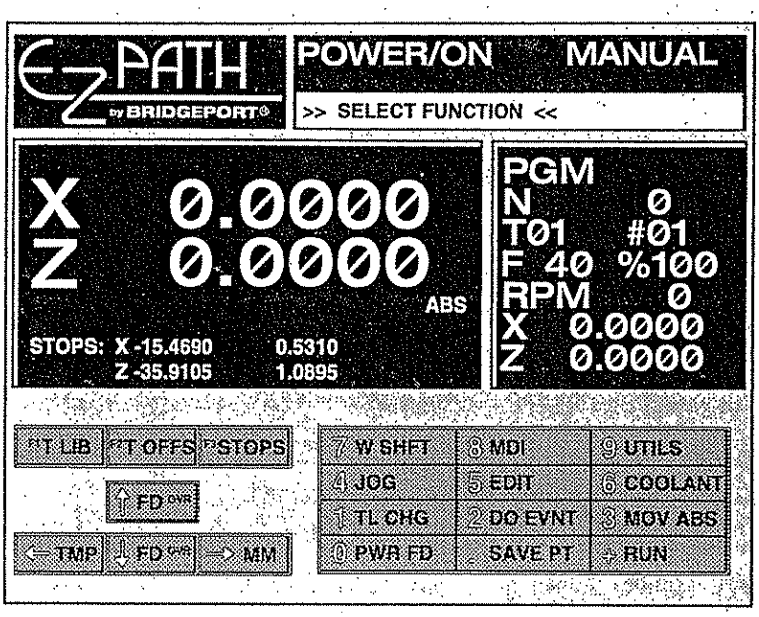


Figure 2-2

The screen shown in Figure 2-2 is the BASIC OPERATIONS screen and is displayed when the EZ-PATH is first powered up. Refer to Appendix D for more information on the X and Z coordinates. To the lower right side of the large XZ display is the indicator for either **Absolute** or **Temporary** mode. This indicator is either given as **ABS** or **TMP**. The screen shown above is in **Absolute** mode. When in **Absolute** mode, the left arrow cursor key has the function of shifting to **Temporary** mode. When the EZ-PATH is in **Temporary** mode the same key has the function of shifting to **Absolute** mode.

The Temporary mode is an alternate coordinate system that allows the user to break into a job to perform a second job without effecting the primary job setup. It allows tools to be set one at a time. In this mode the tool offsets are disabled.

When the power switch is turned on, the EZ-PATH defaults to the **Absolute** mode.

To the right of the large XZ display is a smaller box with a similar set of XZ coordinates at the bottom. This area gives the operator several important pieces of information.

PGM This gives the program number that is currently being run. If no program is loaded into memory this area is blank.

N This shows what line number the EZ-PATH is currently executing. If no program is being executed this area displays a zero (0).

T The tool number that is currently called for in the part program is shown here. The operator is prompted to change the tool when it is necessary.

The tool offset number is shown following the tool number.

F This shows the current feed rate in inches per minute. This value can be overridden by using the feedrate override keys.

% The override percentage of the programmed feedrate is shown here. If the feedrate has been increased using the feedrate override keys, the percentage shown here is greater than 100%. If the feedrate has been decreased using the override keys, the percent shown here is less than 100%.

RPM The spindle speed shown is the **actual** spindle speed, not the programmed value. The actual speed may differ by up to 10%.

X This shows the distance the cross slide has left to travel to reach the target point in the current instruction.

Z This shows the distance the carriage has left to travel to reach the target point in the current instruction.

At the top of the screen, next to the EZ-PATH logo, the power condition of the axis motor drives is shown in large letters. **POWER/ON** means that the axis drives are currently under power. The axes can be jogged using the **JOG** commands, or moved by means of the **MOVE ABS** command, or by turning the handwheels.

The **POWER/OFF** condition is shown only if the axes have not been homed. The EZ-PATH defaults to the **POWER/OFF** condition when the power switch is first turned on.

Directly to the right of the axis power condition is the operational mode display. In Figure 2-2 the **MANUAL** mode is shown. In the **DO EVENT** mode this area shows **OPER CMD** to show that the machine is ready for the operator's command. When a program is running, this display shows **SET: RUN** to indicate that the program is set and is in the **RUN** mode.

The lower half of the display shows the operator's keyboard and the functions of the keys on the keyboard. The keys shown on the screen appear in the same way that they are arranged on the keyboard. The functions of the keys will change when the machine changes from one mode to another. It is important to pay close attention to the functions of the keys as they are shown on the screen.

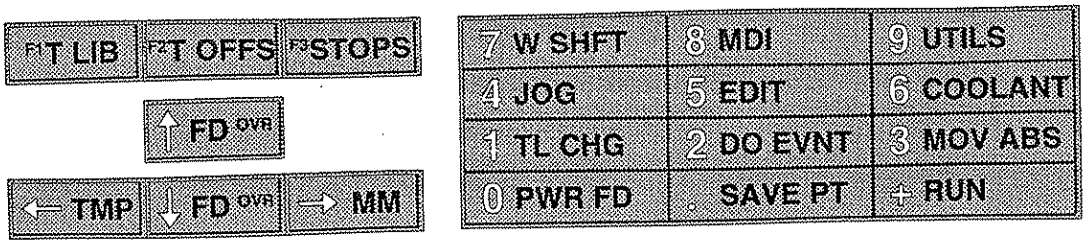


Figure 2-3

Figure 2-3 shows the keyboard as it is shown on the EZ-PATH's start-up screen (see Figure 2-2). The key functions shown here are different from the key functions shown in the DO EVENT mode\ or the EDIT mode. Each of the chapters in this manual shows and describes the appropriate key functions in detail. For an explanation of the keys shown in Figure 2-3, see chapter 4, BASIC OPERATIONS.

CHAPTER 3
EZ-PATH TUTORIAL

Introduction

This section of the manual describes in detail, the basic operation of the machine, by means of a step-by-step tutorial. It is a good idea to read through this section of the manual first, before beginning any operation on the EZ-PATH.

Cutting a Part on the EZ-PATH

This tutorial describes cutting a simple part on the EZ-PATH using the **DO EVENT** mode to execute instructions one at a time. Each programmed event is given in this tutorial with a brief explanation. This tutorial assumes that you have already read the descriptions in Chapters 1 and 2, and that you are familiar with the basic operation of the EZ-PATH.

This tutorial also assumes that you have some experience operating a manual lathe, or you are at least familiar with the operation of one.

The blueprint for the part you are going to cut is shown below in Figure 3-1.

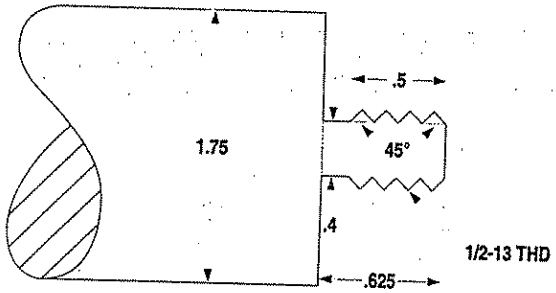


Figure 3-1.

Under normal conditions, this part would not be cut using the DO EVENT mode. Instead, a part program to cut this part would be written using the **MDI** mode, and then executed using the **RUN** mode. However, to demonstrate the EZ-PATH and use of the machining commands, we will step through cutting this part one instruction at a time.

It is highly recommended that you read through this tutorial completely before attempting to cut this part on the EZ-PATH.

Beginning the part

1) Turn on the EZ-PATH. Do this by turning the MAIN DISCONNECT on the front of the cabinet to the **ON** position.

2) Home the axes by pressing the [3] key (**MOV ABS**) on the operator's keyboard.

3) Press the [+] key to execute the **HOME AXES** routine.

4) Based on the blueprint in Figure 3-1, select your work material and tools. The tools should be an OD turning tool, an OD grooving tool and a 60° threading tool. The suggested material is 1 3/4" diameter aluminum bar.

5) Secure the work material in the spindle so that there is NO LESS THAN 2" hanging out.

6) Insert your first tool (OD turning tool) in the tool holder and secure it. If you are using a turret holder, insert all the tools into the holder and secure them (be sure the tools are balanced in the turret).

7) Set the spindle speed at **1060 RPM** using the chart on the front of the machine and positioning the handles to the corresponding colors or numbers.
   Set the spindle speed switch to **High**; set the I II III handle to **I**; set the Red/Blue handle to **Blue** and set the Green/Yellow handle to **Yellow**.
   OR
   Set spindle speed switch to **Low**; set the I II III handle to **III**; set the Red/Blue handle to **Blue** and set the Green/Yellow handle to **Yellow**.

Setting the Tool Offsets

1) Press the [F2] **T OFFS** key. A screen as shown in Figure 3-2 will appear.

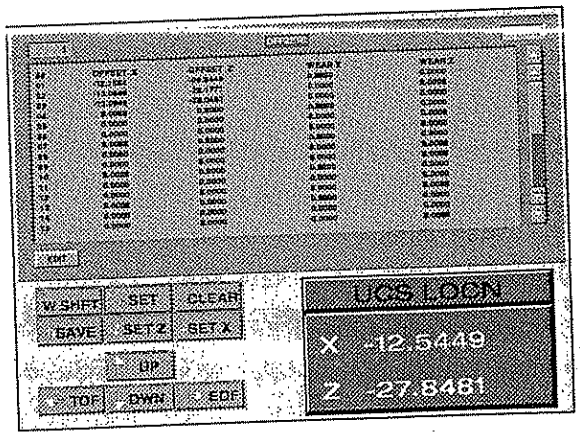


Figure 3-2

2) Use the [↓] **(down arrow)** key to select the proper offset number.

3) Jog the tool to the work piece (manually use the hand cranks or use the power jog knob on the control panel).

4) Press the [+] **SET** key. A screen as shown in Figure 3-3 will appear.

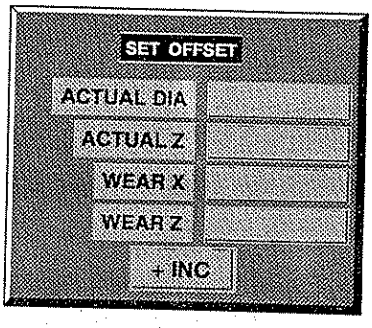


Figure 3-3

5) Manually face the work piece, taking care not to move the Z axis hand wheel when done.

6) With the tool in position, press **<enter>** to get to the ACTUAL Z box. Enter **zero (0)** into the **ACTUAL Z** box and press **<enter>** three times to return to the offsets screen.

7) Press the [+] **SET** key again.

8) Manually turn down the work piece a few thousandths. Stop the spindle and take a measurement of the cut surface. Take care not to move the X axis hand wheel.

9) Enter the measurement value into the **ACTUAL DIA** box and press **<enter>** four times.

Repeat this process for all the tools. When all the offsets are entered, press the zero [0] **SAVE** key to save all the changes to the offsets. The Basic Operations screen will then return to view.

Roughing the Part

1) Press [1] **TOOLCHG**. Install the turning tool in the holder and select the tool and offset numbers that correspond to the turning tool.
Press **<enter>**, the **Active Tool Menu** will appear as in Figure 3-4.

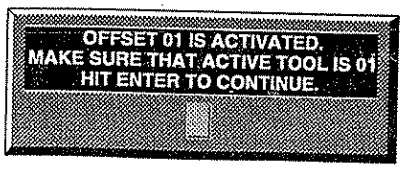


Figure 3-4

2) Press **<enter>** to verify the tool and offset choices.

3) Press the [2] **DO EVENT** key. A screen as shown in Figure 3-5 will appear.

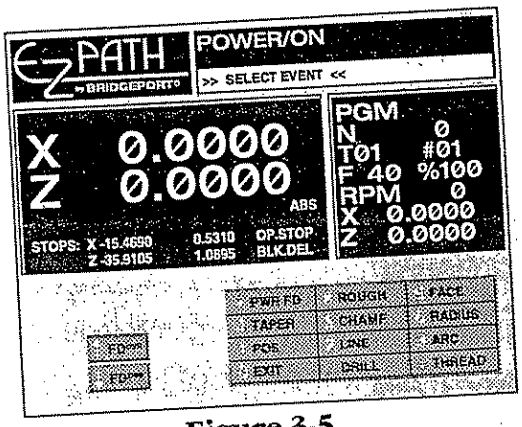


Figure 3-5

4) Position the tool outside the diameter and Z positive from the material.

Press the [8] **ROUGH** key. A screen as shown in Figure 3-6 will appear.

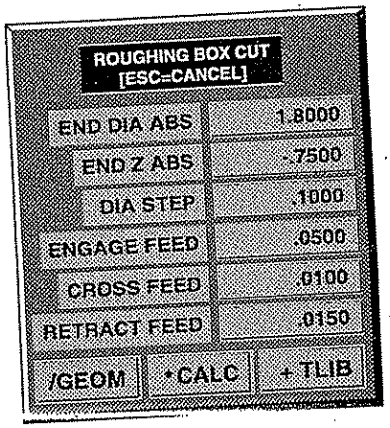


Figure 3-6

5) Enter the values below in the correct spaces. Press the enter key after each number.

END DIA ABS .550 <enter>
END Z ABS -.625 <enter>
DIA STEP .100 <enter>
ENGAGE FEED .020 <enter>
CROSS FEED .010 <enter>
RETRACT FEED .010 <enter>

6) Start the spindle

7) Press the [+] **START** key.

For a finish pass, follow the steps below.

8) Press the [8] **ROUGH** key.

9) Enter the values below in the correct spaces.

END DIA ABS .495 <enter>
END Z ABS -.625 <enter>
DIA STEP 0.000 <enter>
ENGAGE FEED .020 <enter>
CROSS FEED .008 <enter>
RETRACT FEED .010 <enter>

10) Press the [+] **START** key.

11) Press [0] **EXIT** to return to the Basic Operations screen.

When the cutter has ended the finish pass **STOP THE SPINDLE.**

Making the Groove

1) Press the [1] **TOOLCHG** key

2) Install the groove tool in the holder and select the tool and offset numbers that correspond to the grooving tool.

3) Press **<enter>** to verify the tool and offset choices.

4) Set the spindle speed to **530 RPM.**
The settings are **Low, I, Blue, Yellow** OR **High, II, Blue, Yellow.**

5) Manually bring the tool to **Z -.625** and **X DIA 1.9.**

6) Press [F3] **STOPS.** A screen will appear as in Figure 3-7.

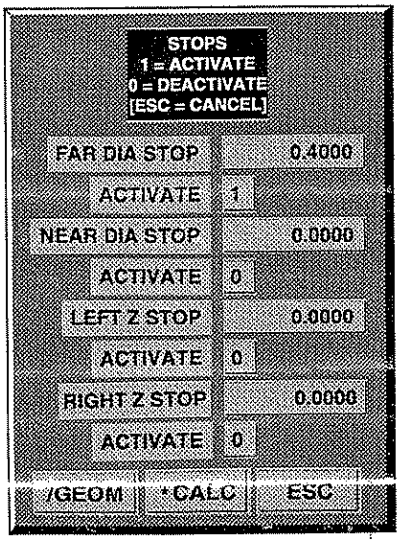


Figure 3-7

7) Enter the following values.
FAR DIA STOP .4 <enter>
ACTIVATE 1 <enter>
Press **<enter>** six (6) times to exit the **STOPS** screen. The Basic Operations screen will appear again.

8) Manually bring the tool to **Z -.625** and **X dia .400** cutting the thread undercut.

9) Press [2] **DO EVENT**. The DO EVENT screen will appear.

10) Press [5] **CHAMF**. A screen as shown in Figure 3-8 will appear.

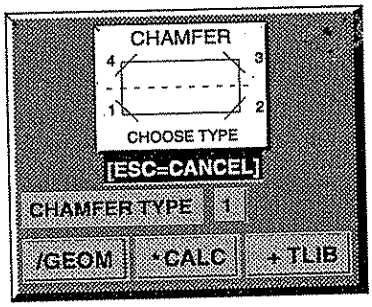


Figure 3-8

11) Choose **type #1 <enter>**.

12) Press [+] **START**.

13) Use the [↑] **up** and [↓] **down arrow** keys to override the feed rate if necessary.

14) Manually turn the **Z** handle **clockwise** to cut the chamfer on the back of the groove.

15) Press [ESC] **escape** twice to end when the groove is finished.

The DO EVENT screen will appear again.

Making the Threads

1) Turn off the stops by pressing the [F3] **STOPS** key. Enter zero **0** in the **ACTIVATE** box.

2) Manually position the tool to **X .350 Z .05**.

3) Press [2] **DO EVENT**.

4) Press [5] **CHAMF**.

5) Choose **type #2 <enter>**.

6) Press [+] **START**.

7) Manually turn the **Z** handle **counter clockwise** to cut the front chamfer.
Press the SELECT button on the operator's panel to toggle to straight positioning.
Reposition the tool to take another deeper cut on the part.
Press the SELECT button on the operator's panel to return to chamfer mode.
Repeat the process of selecting and positioning until the proper chamfer is achieved.

8) Press [ESC] **escape** to end.

9) Press [0] **EXIT** to return to the Basic Operations screen.

10) **STOP THE SPINDLE.**

11) Press [1] **TOOLCHG.**

12) Install the threading tool in the holder and select the tool and offset numbers that correspond to the threading tool.

13) Manually position tool to **X .550 Z .20.**

14) Press [2] **DO EVENT.**

15) Press [+] **THREAD** key. A screen will appear as in Figure 3-9.

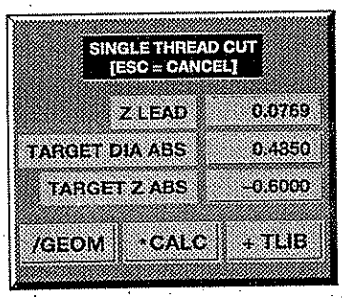


Figure 3-9

16) To find the **Z LEAD** use the calculator function.
   1) Press * **CALC.**
   2) Enter **1/13 <enter>**
      the answer is **.0769** it will be inserted in the correct place.

   TARGET DIA .485 <enter>
   TARGET Z ABS –.60 <enter>

17) . Start the spindle

18) Press the [+] **START** key.

Repeat steps 15 through 18, changing **TARGET DIA** each time by **.01** until **.410** is reached.

For a finish pass repeat steps 15 through 18. Enter **.4040** for **TARGET DIA**.

20) Press [0] **EXIT** to return to the Basic Operations screen.

When the threading operation is complete, **STOP THE SPINDLE**.

The part is now complete.

CHAPTER 4
BASIC OPERATION

Basic operation of the EZ-PATH is controlled from the front panel on the left side of the EZ-PATH and from the front of the machine.

The front panel includes a CRT (Cathode Ray Tube) display for the computer functions of the EZ-PATH, and a keyboard from which commands are given.

The computer screen shows the operator all of the information necessary to run the EZ-PATH. This includes details such as tool location, feedrate, and program information.

The keyboard on this panel is used to enter commands for the EZ-PATH. Each key is assigned a command or function, which is shown on the screen. The function of each key may change as the mode of operation changes. The CRT displays which keys can be used in each mode and the function of each of these keys.

BASIC OPERATION

When power to the EZ-PATH is turned on the BASIC OPERATIONS screen is displayed.

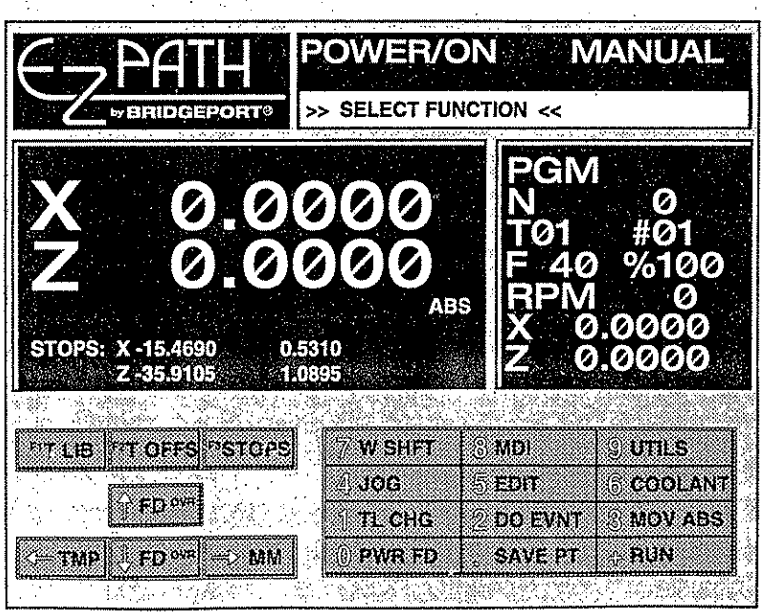


Figure 4-1

From this screen you can access all of the other modes of the EZ-PATH. The screen displays all of the active keys on the keyboard and shows the function of each key. These are explained in this chapter.

To execute a command one of the keys on the keyboard must be pressed. The keys used by the EZ-PATH include numeric keys <**0 – 9**>, math operator keys <**+ – * /**>, cursor keys <**up,down,left,right**>, function keys <**F1–F6**>, <**ESC**>, <**ENTER**> and <**BACKSPACE**>. Note that the function of each key may change depending on the mode of operation of the EZ-PATH.

The key functions shown in the BASIC OPERATIONS screen are:

F1 T LIB. The **F1** key calls the TOOL LIBRARY screen which allows the operator to enter tool parameters, and maintain an accurate tool library. For more information on the TOOL LIBRARY, see chapter 5 in this manual.

F2 T OFFS. The **F2** key allows the operator to enter TOOL OFFSETS and to store these offsets. Each tool should be associated with at least one Tool Offset.

To set a tool offset, a light roughing cut is taken to locate the tip of the tool. The spindle is then stopped and a measurement is made without disturbing the tool's X position. This measured diameter is then entered in the DIA OFFSET box. The Z location is established by facing the part, then selecting the Tool Offset, and entering **0** for the Z offset. Offsets are set in a similar fashion for the rest of the tools being used in the part. Z offsets are set from the face that was established for the first tool.

NOTE: The values entered for **DIA OFFSET** and **Z OFFSET** are not the same as the recorded values in the list of TOOL OFFSETS. This is because the TOOL OFFSETS are the distance from the Machine Coordinate System to the Part Program Coordinate System (Figure 4-2). For this reason it is important to remember which offset is used with each tool. Usually Tool 1 is associated with Offset 1, Tool 2 with Offset 2, etc.

It is also possible to enter WEAR values for X and Z. These values are an incremental adjustment to the main offsets. These values compensate for tool wear and can be reset to zero when the insert or tool is replaced. The values are input with the SET OFFSET dialog box.

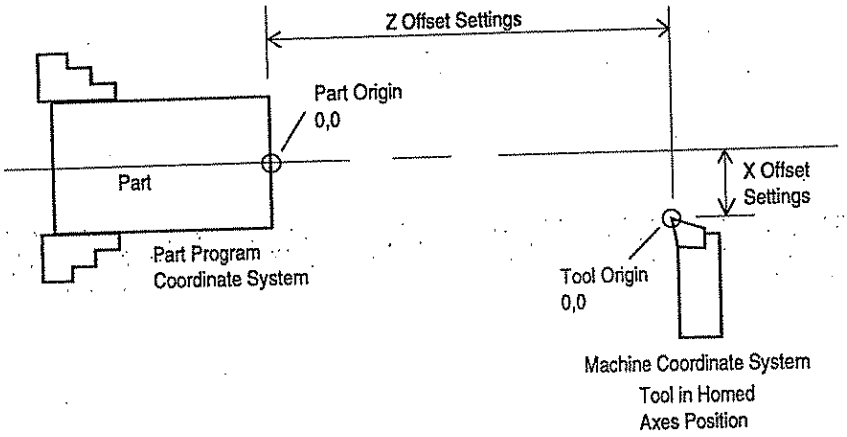


Figure 4-2

In the Tool Offsets screen, use the arrow keys to move the cursor to the Top of File, End of File or Up and Down. Use the + key to **SET** a single offset, the **0** key to **SAVE** all changes, and the * key to **CLEAR** a single offset value. Pressing the **ESC** key exits

the Tool Offsets screen **without** saving any changes. The **/ SET Z** key and the *** SET X** keys are used to quickly enter the offset values for a tool. The dialog boxes for each only allow for the setting of the **ACTUAL X** or **ACTUAL Z** value. Set **WEAR** values using the **+ SET** key.

The **F1 W SHFT** key command allows the operator to program work shift values in both the X and Z directions in the absolute and incremental modes. Work shift changes or moves all tools in relation to the origin. These shift values allow the operator to move the User Coordinate System (UCS) origin so that the tool is properly aligned with the part. This ensures that the part program is cut in the correct location.

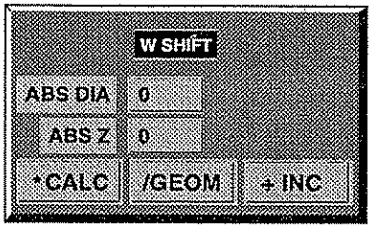


Figure 4-3

The **WORK SHIFT** values set the current position of the tool and carriage to the diameter and Z coordinate values entered in the UCS. For example,

> If the X and Z values are set to **.05** and **0.15** in the WORK SHIFT window, then the UCS origin becomes **.05, 0.15**.

The TOOL OFFSET values set the distance from the Part Program Coordinate System to the User Coordinate System.

NOTE: The main DRO values will reflect the coordinate values input + or - the offset values of the active tool. The UCS display is found in the **F2 T OFFS** screen.

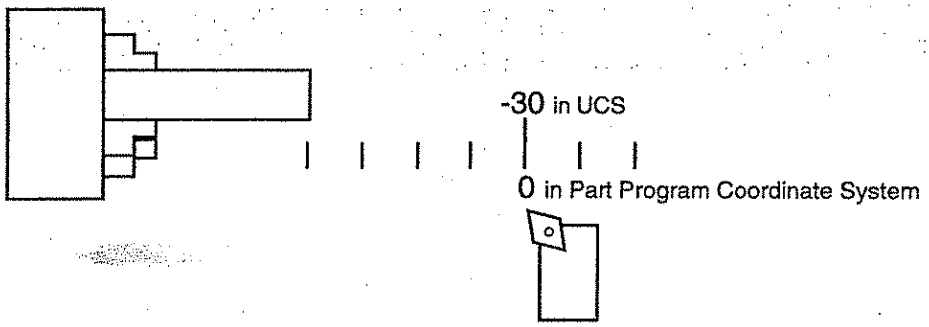


Figure 4-4

For example: the Main DRO reads X 0.000, Z 0.000 and the tool offset for the active tool is Z -30.000 inches (see Figure 4-4). Selecting **WORK SHIFT** and setting Z = +10.000 inches will cause the Main DRO to read X 0.000, Z 40.000 (see Figure 4-5).

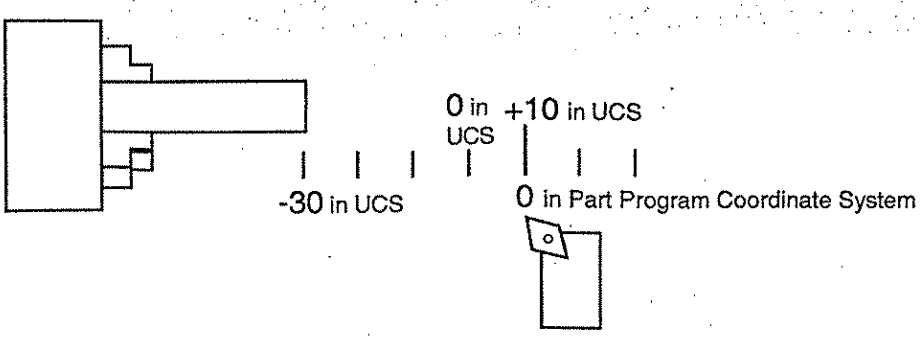


Figure 4-5

F3 STOPS. The **F3** key is used by the operator to program STOP values which limit the travel of the tool. There are four **STOPS** which can be set. These are:

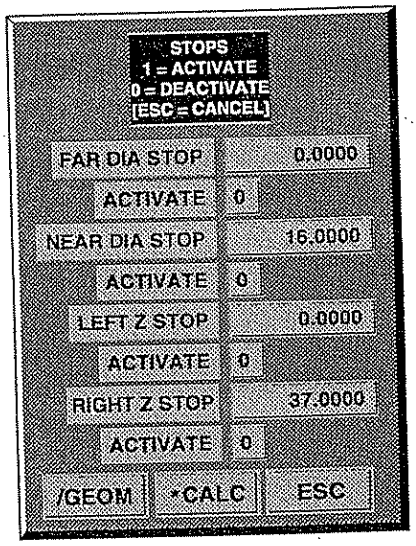


Figure 4-6

FAR DIA STOP **LEFT Z STOP**

NEAR DIA STOP **RIGHT Z STOP**

The STOPS parameters set limits of travel for the tool carriage. The tool may not travel out of the rectangular area that is defined with these values.

FAR DIA STOP sets the minimum diameter value. The tool will not be able to cut into the part beyond this diameter value. This is typically used on an OD operation such as roughing.

NEAR DIA STOP sets a maximum diameter value. The tool is limited to cutting only inside this diameter value. This is typically used on an ID operation such as boring, or roughing.

LEFT Z STOP sets the minimum Z travel limit for the tool. This prevents the tool from cutting too far to the left.

RIGHT Z STOP sets the maximum Z travel limit for the tool. This prevents the tool from cutting too far to the right.

It is not always necessary to have all four STOPS enabled.

STOPS are used to limit the movement of the tool within an operation, such as a command executed in the DO EVENT mode.

For example, the **FAR DIAMETER STOP** might be used to limit a **TAPER** operation. Since this operation is controlled by the handwheels, it is possible for the operator to make incorrect cuts and thus ruin the part.

The FAR DIAMETER STOP, set at the minimum diameter of the taper, would prevent the tool from moving beyond it, and ruining the part. Further examples of using **STOPS** are given in the **DO EVENT** chapter (chapter 6) of this manual.

NOTE: Since the **STOPS** values are set in relation to the **X** and **Z** values shown on the screen, these numbers are shifted when a tool change is made, or when the Work Shift values are changed. These numbers may change, but they are kept in relation to the current tool position.

<cursor LEFT> ABS-TMP. This changes the context of the location readout from **absolute** to **temporary**. The current mode is indicated at the lower right corner of the digital readout with an abbreviation either **ABS** or **TMP**.

The **temporary** (TMP) mode is an alternate coordinate system that allows the user to break into a job set up and perform a different job without changing the set up. It does this by resetting the origin location to establish a new coordinate origin. To do this, select the **TMP** mode, then press the **SET XZ** key and enter **0** for both the **X** and **Z** values. The large DRO display now reads **0.0000** for both **X** and **Z**. The DRO will show the tool location relative to this point until the **ABS** mode is selected again. If SET XZ is not used to set a new origin, the TMP mode will be no different than the ABS mode. If **TMP** is selected a second time, the coordinates are still shown relative to the previous origin settings.

<cursor RIGHT> INCH-MM. This switches the measuring system from INCH to METRIC. The current measurement mode is indicated by the position of the decimal point in the digital display. If three decimal places are shown, the mode is metric. Four decimal places are shown in inch mode. If the machine is in metric mode, all distances are in millimeters and feed rates are in millimeters per revolution.

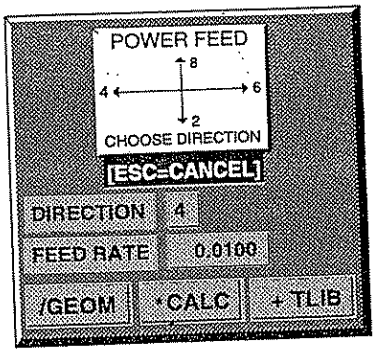
NOTE: To write a program using the MDI mode in **metric**, select the **MM** (right cursor arrow) before selecting the **MDI** command.

<cursor UP>
<cursor DOWN> FDR^{OVR} The FD^{OVR} commands are used to override the programmed feedrate. The <up arrow> key raises the feedrate by 5% each time it is pressed. The <down arrow> key lowers the feedrate by 5% each time it is pressed. The feed override amount is shown as a percent of the programmed feed rate on the right side of the screen.

<.> SAVE PT. This enables the operator to select up to 100 points, which are stored in the system. The points can then be replayed to replicate the same sequence of moves the operator originally went through to machine a part. For more information on the **SAVE PT** commands see chapter 11 in this manual.

+ RUN. The + key in the BASIC OPERATIONS screen calls the **RUN** mode. The **RUN** mode is used to execute or to preview a part program. See chapter 9 in this manual for more information.

0 PWR FD - The **PWR FD** (POWER FEED) command moves the tool in the selected direction at the specified feed rate until the operator presses the **ESC** key or a stop is reached.



DIRECTION — The **DIRECTION** sets which way the tool will move. The direction is chosen according to the four arrows shown on the keyboard. Valid directions are **2**, **4**, **6**, or **8**.

FEED RATE — The **FEED** sets the speed at which the tool moves.

NOTE: The tool will move as soon as the **START** key is pressed. To stop the tool movement, press the **ESC** key.

Figure 4-7

1 TL CHG. The **TOOL CHANGE** command allows the operator to choose a different tool from the tool offset screen. The operator may also view the different programmed tools in the library within this command. The tool number parameter ranges from 1 to 24. These are the only valid values that can be entered here.

2 DO EVNT. This command calls up the DO EVENT screen which allows the machine to execute programmed instructions one at a time. See the chapter entitled **DO EVENT** for more information on the DO EVENT mode and its commands.

3 MOV ABS This command moves the tool to a new point. The coordinates of, or distance to, the new point must be entered on the screen before the tool is moved. The current tool position coordinates appear automatically in this screen. Use this command (instead of the JOG command) to move the X and Z axes at the same time to a position where the DIA and Z coordinates are known. The **MOV ABS** command is also used to **HOME THE AXES** if this has not been done already. The DIA and Z coordinate values can be entered directly, or the **GEO** or **CALC** sub commands can be used . Each of these two commands makes the system calculate the DIA and Z coordinates based on other data entered at the keyboard.

NOTE: The EZ-PATH expects that the X axis is positioned using DIAMETER values. This applies to all modes of the EZ-PATH operation.

CALC

The **CALC** command can be used to enter and evaluate mathematical equations so that points can be calculated. Trigonometric functions, square roots, and exponential functions can be used in the entered equations.

GEO

The **GEO** command can be used to enter other geometry data, such as coordinates, angles and radius values to calculate point locations for machining operations.

4. **JOG**. This command is used to move the X or Z axis, by pressing the **JOG BUTTON** or by using the **STEP** + and – keys. Selecting **JOG** will call up the JOG SCREEN. Use this command (instead of the MOVE commands) to move the Z axis separately from the X axis, or if the coordinates of the desired position are not known (e.g. You want to move the tool to the left side of the part instead of the right side). When the **JOG** mode is selected the screen displays:

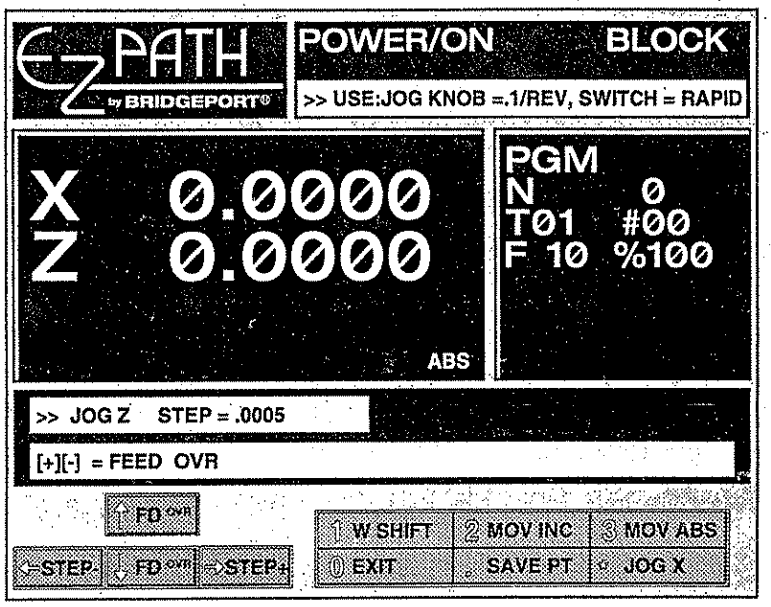


Figure 4-8

<cursor LEFT> STEP – Pressing this key in **JOG** mode will jog the axis (either X or Z) by **–0.0005** inches. This key always jogs the axis in the negative direction.

<cursor RIGHT> STEP + Pressing this key in **JOG** mode will jog the axis (either X or Z) by **+0.0005** inches. This key always jogs the axis in the positive direction.

<cursor UP>
<cursor DOWN> FDR^{OVR} The FD^{OVR} commands are used to override the programmed feedrate. The <up arrow> key raises the feedrate by 5% and the <down arrow> key lowers the feedrate by 5% each time they are pressed. The feed override amount is shown as a percent of the programmed feed rate on the right side of the screen.

0 EXIT Pressing the **0** key exits the **JOG** mode and returns to the BASIC OPERATIONS screen.

1 W SHIFT The **W SHFT** command allows the operator to program work shift values in both the X and Z directions in the absolute and incremental modes. These shift values allow the operator to move the Machine Coordinate origin so that the tool is properly aligned with the part. This ensures that the part program is cut in the correct location. For a complete explanation of **W SHFT**, see the section of this chapter on Tool Offsets.

2 MOV INC The **MOV INC** command in the **JOG** mode works the same as the **MOV INC** command in the BASIC OPERATIONS mode. It prompts the user to enter distances to move in each direction.

3 MOV ABS The **MOV ABS** command in the **JOG** mode works the same as the **MOV ABS** command in the BASIC OPERATIONS mode. It prompts the user to enter a set of coordinates to which the tool is then moved.

. SAVE PT The **SAVE PT** command in the **JOG** mode works the same as the **SAVE PT** command in the BASIC OPERATIONS mode. It allows the operator to store up to 100 points and replay these as positioning or turning events.

*** JOG X** or **JOG Z** The * key in the **JOG** mode is labeled as either JOG X or JOG Z. This key switches from the current JOG screen to the JOG screen for the other axis (whichever axis is not currently active).

5 **EDIT**. The **EDIT** command enables the **EDIT** mode. This is used to edit a part program which was saved previously. For more information on the **EDIT** mode, see the chapter in this manual called **EDIT**.

6 **COOLANT**. This command is used to turn the flood coolant on or off.

Figure 4-9

7 **W SHFT**. The **7 W SHFT** command is the same as the **F1 W SHFT** as described with **F2 T OFFS**. For a complete explanation of **W SHFT**, see the section of this chapter on Tool Offsets.

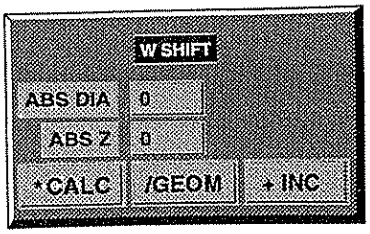
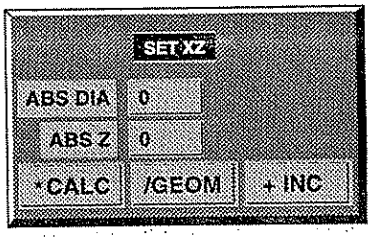


Figure 4-10



If the TMP mode is selected, the **7** key will read **SET XZ**. If this key is pressed, the screen will look as shown in Figure 4-8. This command functions the same way in the TMP mode as Work Shift does in the ABS mode.

Figure 4-11

8 **MDI**. The **MDI** mode provides the ability to create PART PROGRAMs. For more information on the **MDI** mode, and the corresponding commands available, see the chapter in this manual called **MDI**.

9 **UTILS**. The **UTILS** key in the BASIC OPERATIONS screen temporarily exits the EZ-PATH environment to allow the user to run the disk utility program which is used to copy programs, copy disks, and manage files. A utility is also provided for sending files to, and receiving files from, a remote computer. For more information on the **UTILITIES**, see chapter 10 in this manual called **UTILITIES**.

CHAPTER 5
THE TOOL LIBRARY

INTRODUCTION

The EZ-PATH has a library of tools which are utilized in part programs that are created in the MDI mode. In a program, several different tools may be required to complete a part, so it is necessary to give each tool an ID number. The Tool Library stores the parameters which define each tool and saves these parameters with the appropriate ID number.

NOTE: The tools in the Tool Library are called by their ID number. The **TL CHG** operation in the DO EVENT mode calls a **tool number** referring to the tool position in the optional tool turret. These are **not** the same.

The **T LIB** (Tool Library) mode is used to add or delete tools, or to edit the parameters which make up a tool entry in the Tool Library. The Tool Library mode is called from the EZ-PATH Basic Operations screen by pressing the **F1 T LIB** key. A listing of the Tool Library is then displayed on the screen.

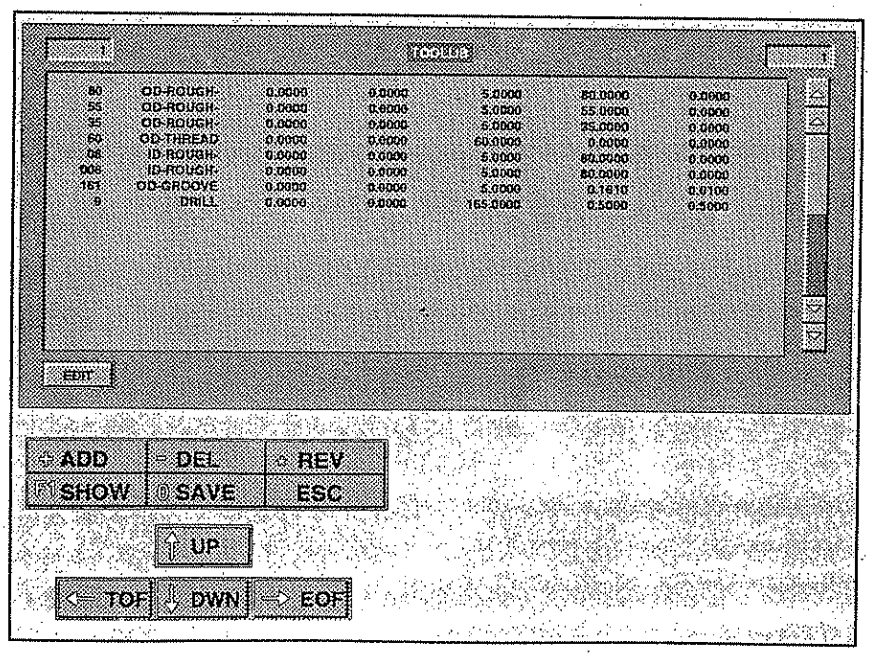


Figure 5-1

From this screen, tools can be displayed, added, deleted, or revised. The cursor arrow keys are used to select a tool. Each tool in the Tool Library is assigned a unique ID number that is six (6) characters or less. This allows the EZ-PATH to identify each tool quickly, and to save the programmed parameters of each tool.

Adding a New Tool

When the **ADD** command is selected from the Tool Library, the screen will display:

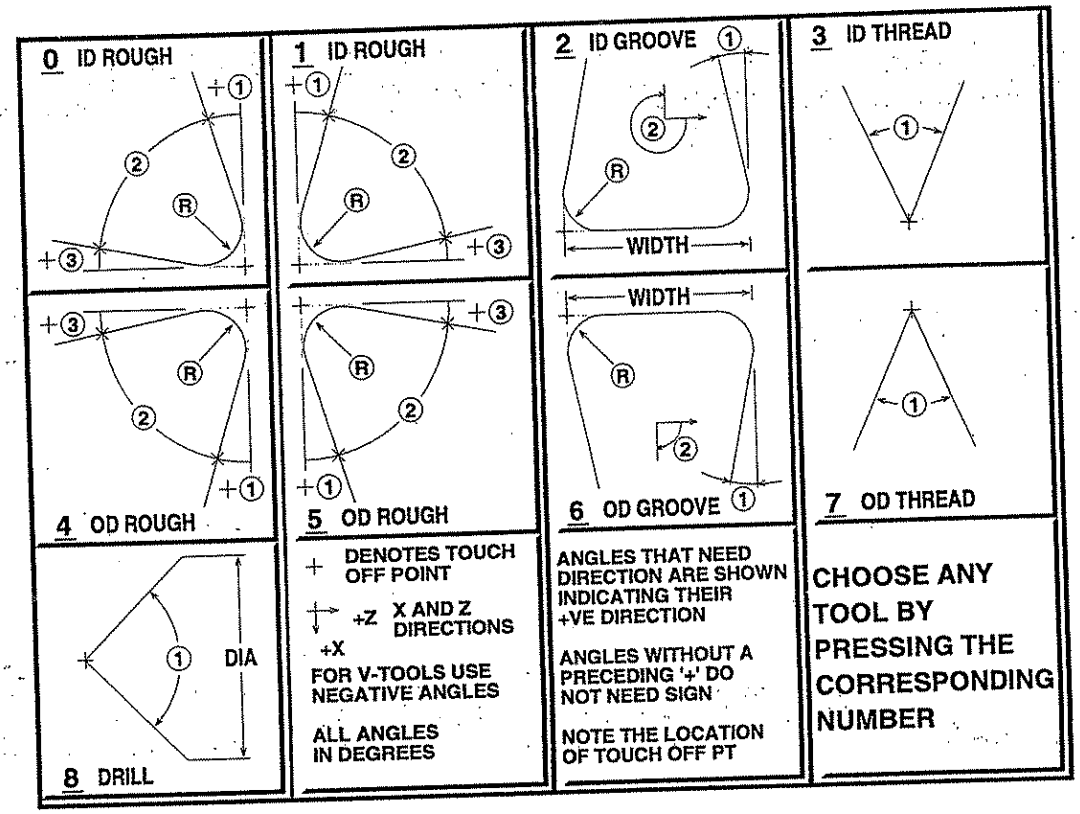


Figure 5-2

There are nine types of tools that can be programmed into the EZ-PATH. Each of the nine tool types has its own parameters. Every tool should be entered according to its type, and each tool parameter should be checked carefully. Incorrect tool parameters can cause damage to the tool, the part, and the machine.

Tool Parameters

Most of the numbers which define a tool are supplied with the tool insert itself, or from a tool book which supplies dimensions for different types of tools. Different tool types have different dimensions. Drills and threading tools are not specified with the same parameters.

DIST TO TOUCH OFF

The EZ-Path has to have one point on the tool which it considers to be the location of the tool. This point is denoted by the cross hairs in the diagram. Under normal operation this is the **touch off** point. The touch off point is found by touching the tool to the part blank and then setting the X and Z locations. In most cases the **DIST TO TOUCH OFF X and Z** values can be left at zero (0). If it is necessary to move the touch off point, it is done by changing the X and Z values for the distance to touch off. The cross hairs will be shown in the new location.

In some cases it may be desirable to program the **tool center** (or some other location) instead of the touch off point. This is done by entering values for the **DIST TO TOUCH OFF X AND Z** parameters.

It is possible to program the EZ-PATH to do turning with square (grooving type) tools. This is done by defining the tool shape as a Grooving Tool and calling for that tool in the part program.

Tool Types

0 ID ROUGH This tool type is an ID turning tool that cuts in the positive Z direction.

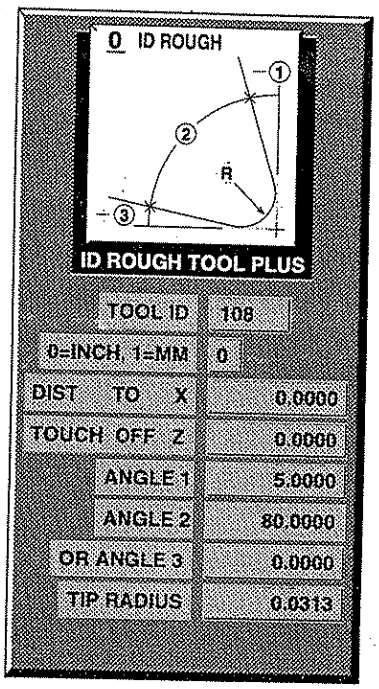


Figure 5-3

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool must be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the relief angle from the X axis.

ANGLE 2 This is the **included angle** of the tool.

ANGLE 3 This the relief angle from the Z axis. It is only necessary to enter two of the three angles.

TIP RADIUS This is the radius of the tool's tip.

1 ID ROUGH This tool type is an ID turning tool that cuts in the negative Z direction.

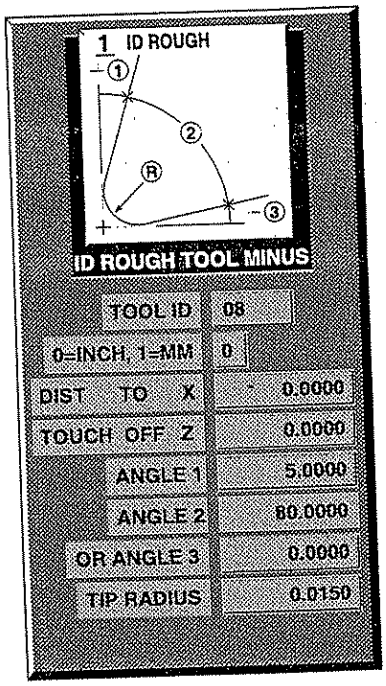


Figure 5-4

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the relief angle from the X axis.

ANGLE 2 This is the **included angle** of the tool.

ANGLE 3 This the relief angle from the Z axis. It is only necessary to enter two of the three angles.

TIP RADIUS This is the radius of the tool's tip.

2 ID GROOVE This tool type is an ID grooving tool.

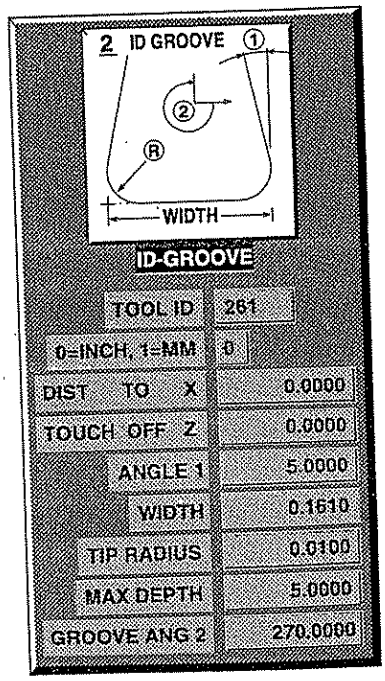


Figure 5-5

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the relief angle from the X axis.

WIDTH This is the **total** width of the tool.

TIP RADIUS This is the radius of the tool's tip.

MAX DEPTH This is the greatest depth to which the tool can cut.

GROOVE ANG 2 This is the orientation of the groove on the part. (i.e.: ID groove, face groove, etc.)

3 **ID THREAD** This is an ID threading tool. This tool is used to cut threads on the inner diameter of the workpiece.

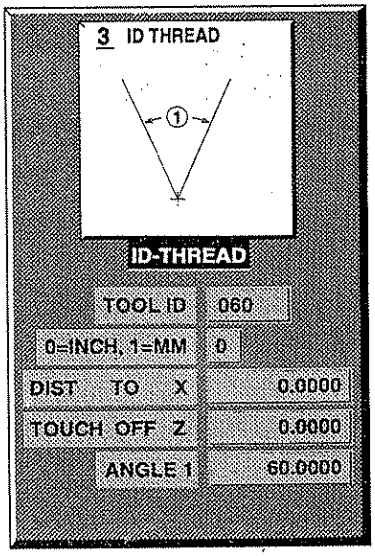


Figure 5-6

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the included angle of the tool.

4 **OD ROUGH** This tool type is an OD turning tool that cuts in the positive Z direction. This tool type may be selected for a turning, or facing tool.

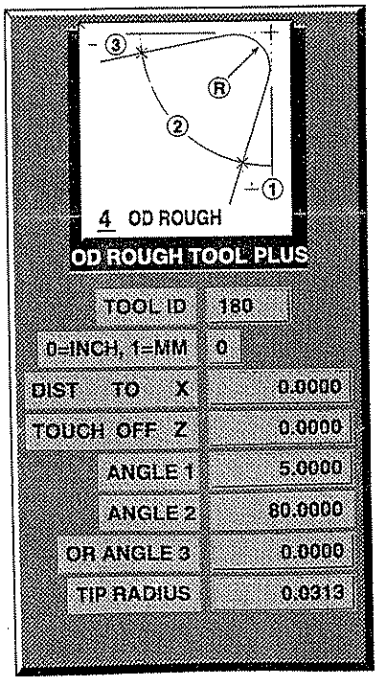


Figure 5-7

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the relief angle from the X axis.

ANGLE 2 This is the **included angle** of the tool.

ANGLE 3 This the relief angle from the Z axis. It is only necessary to enter two of the three angles.

TIP RADIUS This is the radius of the tool's tip.

5 OD ROUGH This tool type is an OD turning tool that cuts in the negative Z direction. This tool type may be selected for a turning, or facing tool.

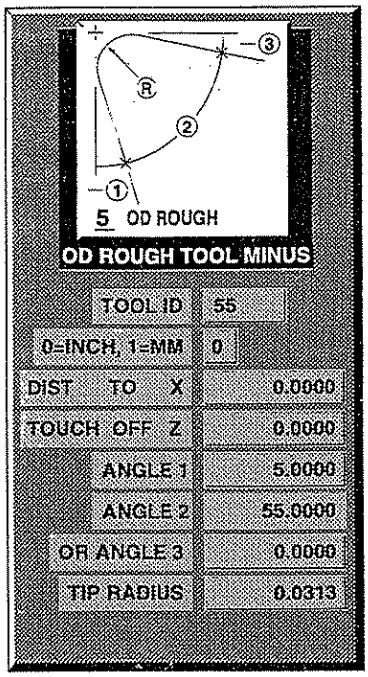


Figure 5-8

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the relief angle from the X axis.

ANGLE 2 This is the **included angle** of the tool.

ANGLE 3 This the relief angle from the Z axis. It is only necessary to enter two of the three angles.

TIP RADIUS This is the radius of the tool's tip.

6 OD GROOVE This tool type is an OD grooving tool.

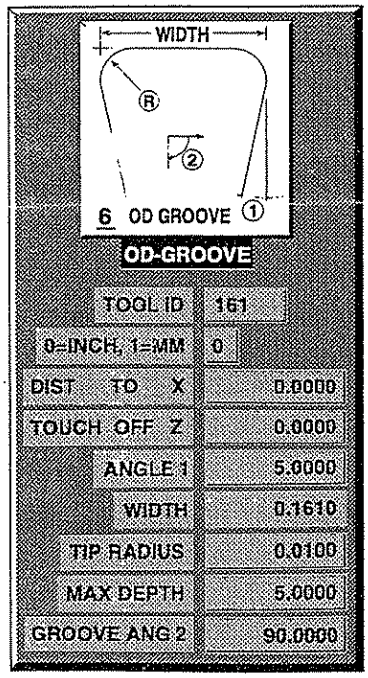


Figure 5-9

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the relief angle from the X axis.

WIDTH This is the **total width** of the tool.

TIP RADIUS This is the radius of the tool's tip.

MAX DEPTH This is the maximum depth that the tool can cut.

GROOVE ANG 2 This is the orientation of the groove on the part. (i.e.: ID groove, face groove, etc.)

7 **OD THREAD** This tool is an OD threading tool. This tool type is used to cut threads on the outer diameter of the workpiece.

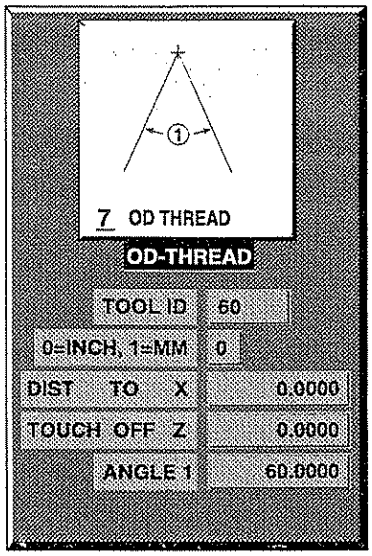


Figure 5-10

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the included angle of the tool.

8 **DRILL** This is a drilling tool.

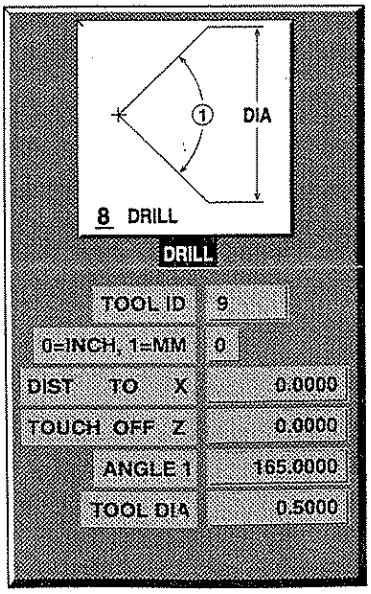


Figure 5-11

TOOL ID This is the ID number that is used to select a tool from the Tool Library. Each tool must have a unique ID number.

0=INCH, 1=MM A tool can be programmed in either metric or inch mode.

DIST TO X
TOUCH OFF Z These values give the distance from the touch off point to a different program point (e.g. the tool center).

ANGLE 1 This is the included angle of the drill point.

TOOL DIA This is the **diameter** of the tool.

Deleting a Tool

To delete a tool from the Tool Library list, move the cursor (highlighted) to the tool to be deleted by using the cursor arrow keys. Press the – key to delete this tool. A prompt appears asking **CONFIRM DELETE TOOL [0] = OK**. Press zero and then press the **Enter** key to delete the tool.

Revising a Tool

To change the parameters of any tool in the Tool Library, place the cursor (highlighted) on the desired tool. Press the * **REV** key to display the data entry screen for that tool. Changes are updated when all of the parameters are entered. To save changes to the Tool Library file on the disk the **0 SAVE** key must be pressed.

Displaying a Tool

Any tool currently in the Tool Library may be viewed. The programmed parameters are used to show a graphic approximation of the tool shape. This tool shape may not exactly reflect the shape of the actual tool. To view a tool, select the tool by placing the cursor (highlighted) on the desired tool, using the cursor arrow keys. Press the **F1 SHOW** key to display the tool tip. The tool is displayed for several seconds. If the cursor is positioned on the first line (ID . TOOL TYPE etc) all the tool assigned will be displayed until the **Enter** key is pressed.

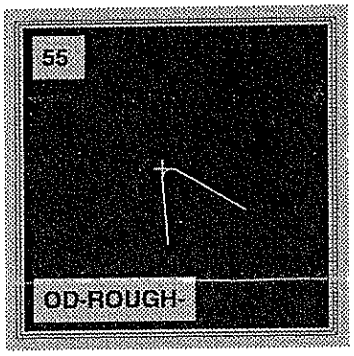


Figure 5-12

Saving the Tool Library

The Tool Library is saved as a file on the EZ-PATH hard disk. When any changes are made to the Tool Library select the **0 SAVE** key to save the changes to this file.

Quitting the Tool Library

The Tool Library remains on the screen until the Tool Library file is saved, or until the **ESC** key is pressed. If the **ESC** key is pressed, the system warns the user that all changes to the Tool Library will be abandoned. If the user acknowledges the warning, the screen returns to the Basic Operations screen.

CHAPTER 6
DO EVENT OPERATION

Introduction

The EZ-PATH has a DO EVENT mode in which instructions can be programmed and executed one at a time. This means that simple automatic operations such as turning, facing or threading, can be executed with a single command instead of being performed manually. Cutting a chamfer, taper or radius are done with the semi-manual modes. All of the EZ-PATH's DO EVENT cycles are described in this chapter. When the DO EVENT mode is selected, the display changes.

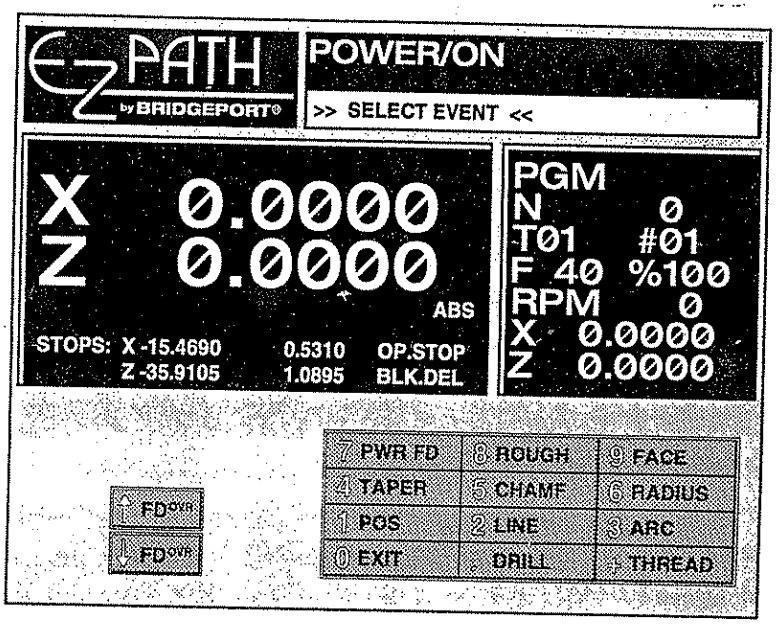


Figure 6-1

NOTE: It is important to remember that the EZ-PATH expects **diameter** input for the X axis. In each of the commands in the DO EVENT mode the X axis input is always a **diameter** measurement.

0 EXIT - This command exits the DO EVENT mode and returns the display to the BASIC OPERATIONS screen.

. **DRILL** - The **DRILL** command is used to drill a hole at the center of the part. The hole can be drilled in "steps" or pecking depths, to clear chips and prevent overheating either the drill or the part. The drill will move to the center line automatically.

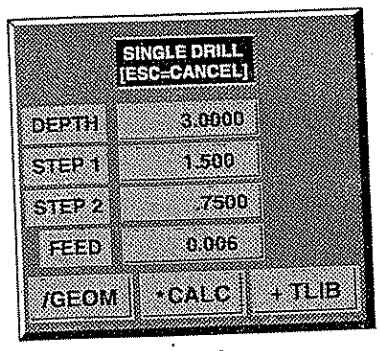


Figure 6-2

DEPTH The **DEPTH** value gives the distance the tool will feed into the workpiece from the current tool position.

STEP 1 The **STEP 1** value sets the depth of the first peck.

STEP 2 The **STEP 2** value sets the depth of the second peck and all following pecks.

FEED The **FEED** value sets the feed rate for the entire drilling operation. This value is in inches per revolution.

For example, the values shown in Figure 6-2 would produce this DO EVENT line:

```
0000 DRILL D3.0000 1.5000 0.7500 F0.0060
```

When the **START** key is pressed, this command produces the tool movement shown in Figure 6-3.

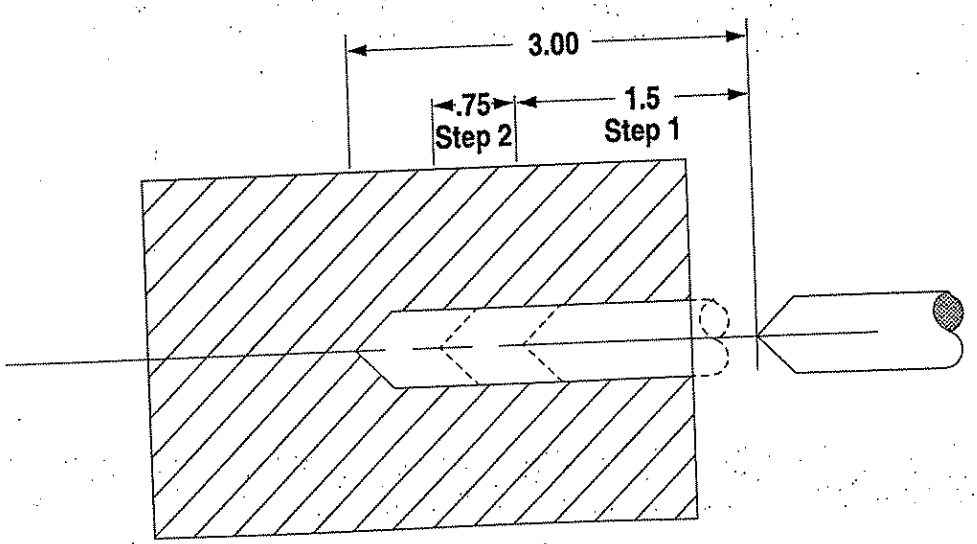


Figure 6-3

+ THREAD - This command cuts threads either on the inside or the outside diameter of the part. The movement of the axes is controlled by the computer.

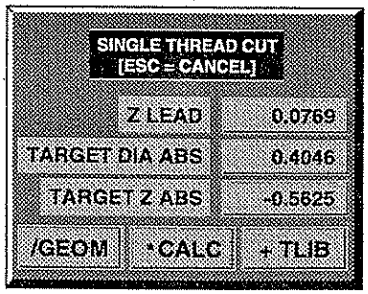


Figure 6-4

Z LEAD The **Z LEAD** value is the distance from the top of one thread to the next thread. This is calculated as 1/(threads per inch).

TARGET DIA ABS This parameter gives the depth of the thread cut. Threads are cut with multiple passes, it is necessary to change this value for each cutting pass.

TARGET Z ABS The **TARGET Z** is the absolute end point of the thread cutting pass. The tool cuts to this point, then withdraws by the clearance height, and rapids back to the start position.

For example:

The values shown in Figure 6-4, above, would create the DO EVENT command:

```
0000 THREAD L0.0769 X0.4046 Z-0.5625
```

Which would produce the threads shown in Figure 6-5 below.

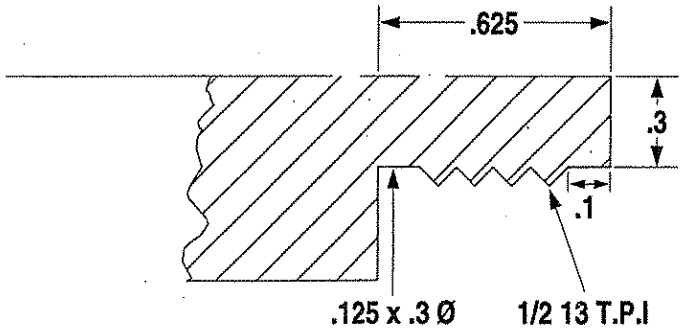


Figure 6-5

1 POS - This command positions the tool at a specific location. The carriage moves at the rate of 100 inches per minute to the new location, so it is important that the tool does not contact the part during a rapid move.

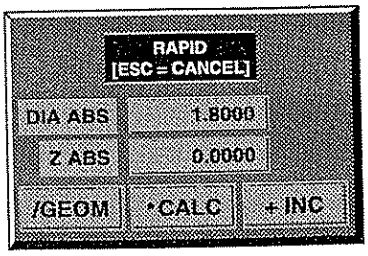


Figure 6-6

DIA ABS The **DIA ABS** value gives the diameter of the point the tool is moving toward.

Z ABS The **Z ABS** value gives the Z coordinate of the point the tool is moving toward.

The **POS** (position) or **RAPID** command can also be programmed in the **incremental** mode. Press the **INC** key, to shift to **incremental** values. These values are distances from the current tool location.

For example:

The values shown in Figure 6-6, above, would create the DO EVENT command:

```
0000 RAPID ABS X1.8000 Z0.0000
```

This command causes the tool movement shown in Figure 6-7 below.

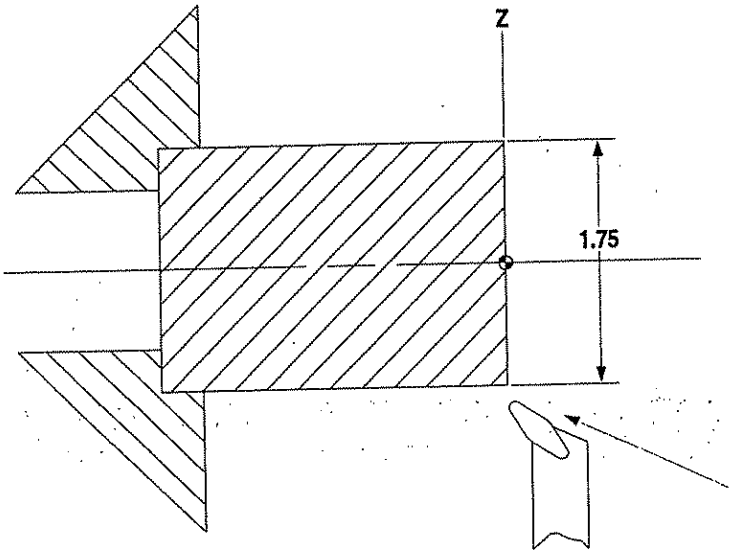


Figure 6-7

2 LINE - This command moves the tool in a straight line from the current position to the position that is entered as a Z coordinate and diameter. The tool is moved at the specified feedrate, cutting the part as it moves.

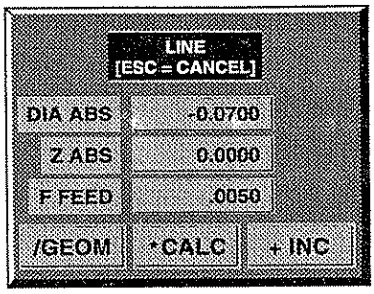


Figure 6-8

DIA ABS The **DIA ABS** value gives the diameter of the point the tool is moving toward.

Z ABS The **Z ABS** value gives the Z coordinate of the point the tool is moving toward.

F FEED The **FEED** value sets the speed of the tool as it moves towards the target point. The feed value is in inches per revolution.

The **LINE** command can also be programmed in the **incremental** mode. Press the **INC** key, to shift to **incremental** values. These values are the distances the tool will move from the current tool location.

For example:

The values shown in Figure 6-8, above, would create the DO EVENT command:

```
0020 LINE ABS X-0.0700 Z0.0000 F0.0050
```

Which produces the tool motion shown in Figure 6-9 below.

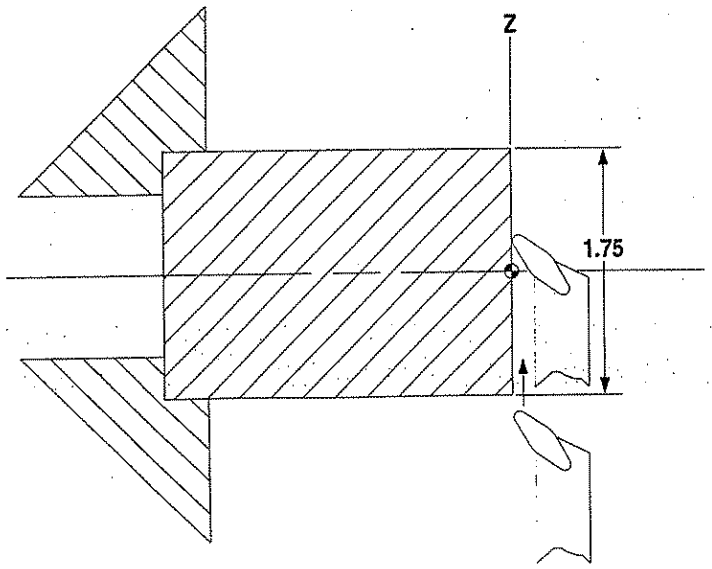


Figure 6-9

3 ARC - This command moves the tool along an arc from the current location, which is assumed to be on the arc, to the position that is entered as a diameter and Z coordinate. The arc is specified by its center point or by its radius.

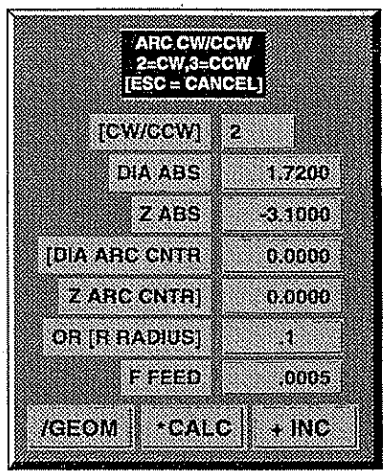


Figure 6-10

NOTE: The tool is assumed to be positioned on the arc before the instruction is executed. The arc (center point or radius) is calculated when the **START** button is pressed.

CW/CCW This value sets the direction of the tool movement on the arc. Enter **2** to cut **clockwise** or **3** to cut **counter clockwise**.

DIA ABS The **DIA ABS** value gives the diameter of the point the tool is moving toward.

Z ABS The **Z ABS** value gives the Z coordinate of the point the tool is moving toward.

[DIA ARC CNTR
ZARC CNTR]
OR [R RADIUS] The arc must be entered either by the **DIA** and **Z** coordinates of its center point, or by its radius value. It is not necessary to enter numbers in all three of these fields.

F FEED The **FEED** value sets the speed of the tool as it moves towards the target point. The feed value is in inches per revolution.

For example:

The values shown in Figure 6-10 would create the DO EVENT command:

```
0030 ARC|RADIUS ABS CW X1.7200 Z-3.1000 R0.1000 F0.0005
```

Which produces the tool motion shown in Figure 6-11.

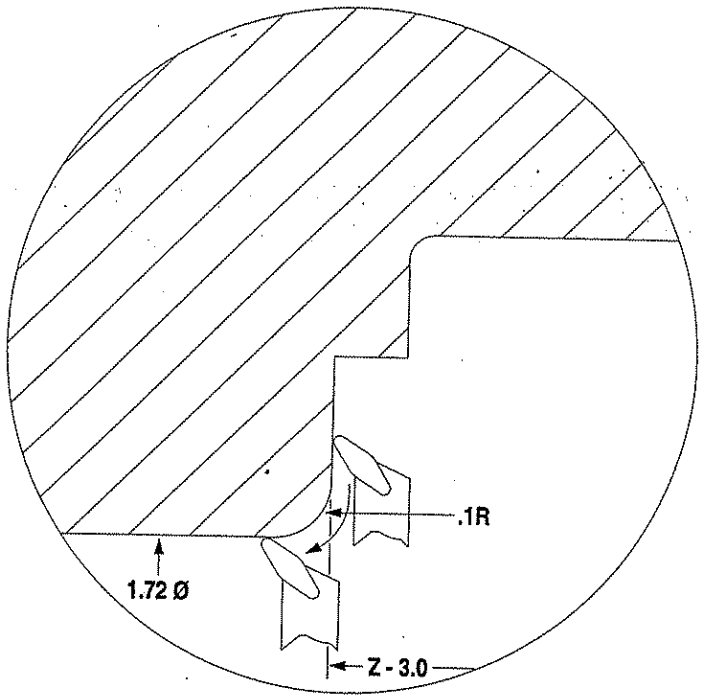


Figure 6-11

4 TAPER - This command cuts a TAPER, which is a gradually changing diameter, in the orientation chosen by the operator. This command slaves one axis drive to the other so that the tool automatically cuts the taper at the specified angle. This command does not move the tool automatically; instead, the X axis is moved in relation to the operator's movement of the Z axis handwheel.

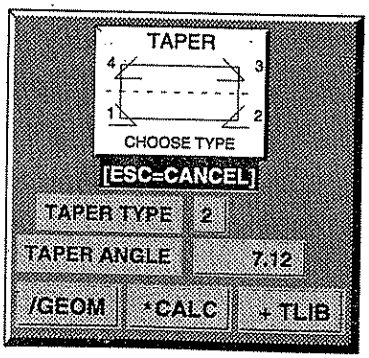


Figure 6-12

TAPER TYPE — The **TAPER TYPE** value sets the orientation of the taper cut. The value entered should be chosen from one of the four orientations shown in the diagram on the screen.

TAPER ANGLE — The **TAPER ANGLE** sets the angle of the taper cut from the horizontal. This is always a positive number.

NOTE: The **TAPER** mode is not in effect until the **START** button is pressed. Once in effect, the **TAPER** mode can be suspended temporarily by pressing the **SELECT** button on the lower control panel on the carriage. This sets the machine in normal manual mode, freeing both handwheels. When **SELECT** is pressed again, the **TAPER** mode is re-activated. To exit the **TAPER** mode, press the **ESC** key.

For Example:

Suppose that we need to cut a **TAPER** like this:

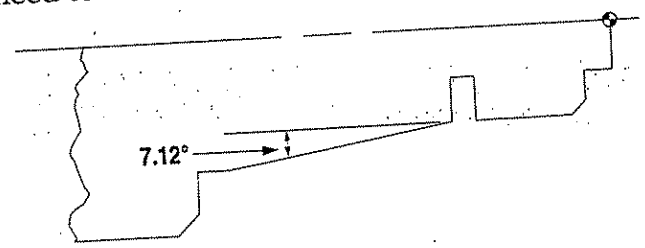


Figure 6-13

To do this, we must make several passes with the **TAPER** mode since there is too much material to be removed in one cut. These cuts will look like this:

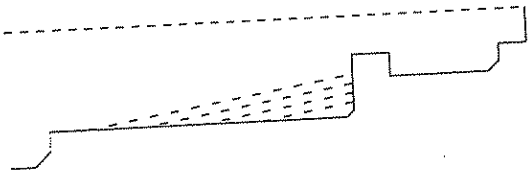


Figure 6-14

The values entered in Figure 6-12 would create the DO EVENT command to cut this part:

```
0000 TAPER T02 7.1200
```

To execute this command, position the tool in the X and Z axes at the groove. Then press the **START** button. The screen will read **TAPER ACTIVE**. The **TAPER** mode is now active.

Move the Z axis handwheel slowly to the left, and the tool moves in both the Z and X axes, cutting the taper at the 7.12 degree angle.

When the first cut is made and the tool clears the part, press the **SELECT** button on the lower control panel to the right side of the carriage. This temporarily deactivates the **TAPER** mode. The screen will read **DUAL JOG MODE**. Both handwheels are now free to jog the tool.

Move the tool back to the right side of the taper, and adjust the carriage to make a second pass. Press the **SELECT** button to reactivate the **TAPER** mode.

This process is repeated until the taper cut reaches the stops that were set. The **TAPER** operation is now complete. Press the **ESC** button to end the **TAPER** operation.

NOTE: The feedrate override command can be used during the **TAPER** mode.

5 CHAMF - This command cuts a 45° chamfer in the orientation specified by the operator. This command slaves one axis to the other so that the tool automatically cuts the chamfer. This command does not move the tool automatically; instead, the X axis is moved in relation to the operator's movement of the Z axis handwheel.

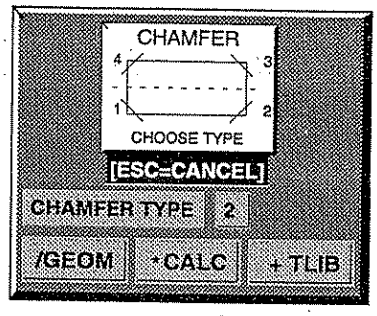


Figure 6-15

CHAMFER TYPE This value sets the orientation of the chamfer cut. The value entered should be chosen from one of the four orientations shown in the diagram on the screen.

NOTE: The **CHAMFER** mode is not in effect until the **START** button is pressed. The screen will display the line **>> CHAMFER ACTIVE**. Once in effect, the **CHAMF** mode can be suspended temporarily by pressing the **SELECT** button on the lower control panel on the carriage. The screen will then display the line **>> DUAL JOG MODE**. This sets the machine in two-axis jog mode, freeing both handwheels. When **SELECT** is pressed again, the **CHAMF** mode is re-activated. To exit the **CHAMF** mode, press the **ESC** key.

For Example:

Suppose that we we need to cut a **CHAMFER** like this:

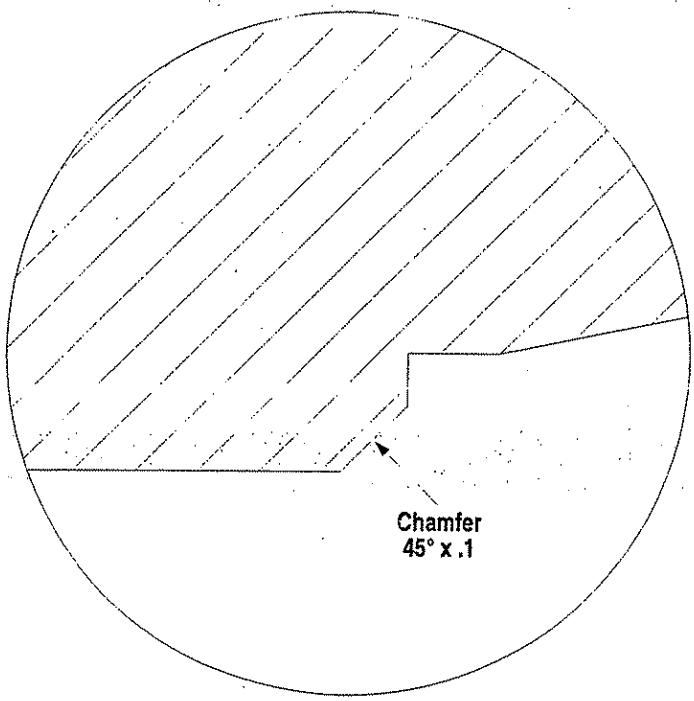


Figure 6-16

This operation can be performed easily using the **CHAMFER** mode, since this is a 45° chamfer.

For this example, we will assume that the zero point is set at the front face of the part and on the center line. Further, we will assume that the **STOPS** have been set as follows:

Far Diameter Stop: 0.50 Left Z Stop: -0.25

These stops will prevent the chamfer operation from cutting more material than is called for in the part design. The tool will stop moving when it reaches that measurement.

First we position the tool in front of the part, just ahead of the material to be removed.

Then, we select the **CHAMF** command and choose orientation **2**, since that is the orientation of this taper.

Now position the tool in the X axis so that it will cut the part when it is moved diagonally to the left. Then press the **START** button. The **CHAMF** mode is now active.

Move the Z axis handwheel slowly to the left, and the tool moves in both the Z and X axes, cutting the chamfer at a 45 degree angle.

When the first cut is made and the tool clears the part, press the **SELECT** button on the lower control panel to the right side of the carriage. This temporarily deactivates the **CHAMF** mode. Both handwheels are now free to move the tool.

Move the tool back to the right side of the part, and adjust the carriage to make a second pass. Press the **SELECT** button to reactivate the **CHAMF** mode.

This process is repeated until the cut reaches the LEFT Z STOP that was set. The **CHAMF** operation is now complete. Press the **ESC** button to end the chamfer operation.

NOTE: The feedrate override command can be used in the **CHAMFER** mode.

6 RADIUS - This command cuts an arc around a specified arc center. This command slaves one axis drive to the other so that the tool automatically cuts a radius. This command does not move the tool automatically; instead, the X axis is moved in relation to the operator's movement of the Z axis handwheel. The radius is programmed by its center point. The net effect is that the tool is moved in concentric circles, allowing multiple cuts around one location.

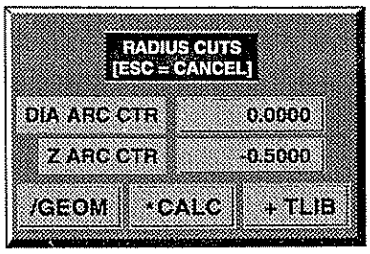


Figure 6-17

DIA ARC CTR The **DIA ARC CTR** is the diameter of the center point of the radius.

Z ARC CTR The **Z ARC CTR** is the Z coordinate of the center of the radius.

NOTE: The **RADIUS** mode is not in effect until the **START** button is pressed. Once in effect, the **RADIUS** mode can be suspended temporarily by pressing the **SELECT** button on the lower control panel on the carriage. This sets the machine in normal manual mode, freeing both handwheels. When **SELECT** is pressed again, the **RADIUS** mode is re-activated. To exit the **RADIUS** mode, press the **ESC** key.

For Example:

Suppose that we need to cut a **RADIUS** like this:

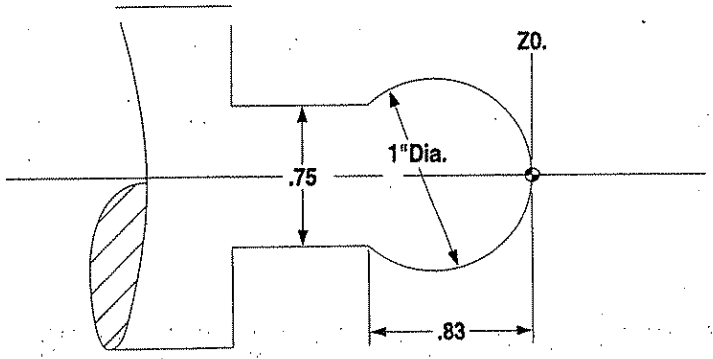


Figure 6-18

To perform this operation we must make several small cuts with the **RADIUS** mode since there is too much material to be removed in one pass.

For this example we will assume that the zero point has been set at the front face of the part on the center line. For this operation the following **STOPS** must be set:

Far Diameter Stop: 0.00 Left Z Stop: -0.83

These values will prevent the **RADIUS** operation from cutting more material than is called for in the part design. The other **STOPS** should be deactivated since they will interfere with the **RADIUS** operation.

First position the tool in front of the radius, just ahead of the front of the part.

Then select the **RADIUS** command and enter **0.0** for the **X ARC CTR** value, and **-0.5** for the **Z ARC CTR** value.

Now position the tool in the X axis, in front of the face of the part. Then press the **START** button. The **RADIUS** mode is now active.

Move the Z axis handwheel slowly to the left, the tool moves in both the Z and X axes, cutting the radius.

It will take several shallow cuts to remove the material necessary to form the radius. After making each cut, press the **SELECT** button on the lower control panel to the right of the tool carriage. This temporarily deactivates the **RADIUS** mode and frees the handwheels to move the tool in both axes.

Move the tool away from the part and back to the front face. Move the tool back into the part and press the **SELECT** button again to reactivate the **RADIUS** mode.

This process is repeated until the radius cut reaches the stops that were set. The **RADIUS** operation is now complete. Press the **ESC** button to end the **RADIUS** operation.

7 PWR FD - The **PWR FD** (POWER FEED) command moves the tool in the selected direction at the specified feed rate until the operator presses the **ESC** key or a stop is reached.

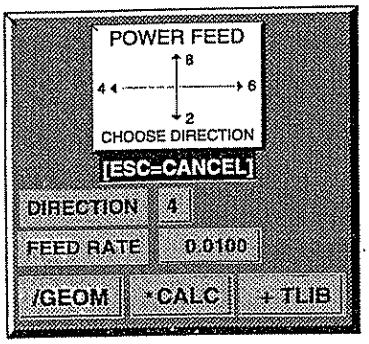


Figure 6-19

DIRECTION — The **DIRECTION** sets which way the tool will move. The direction is chosen according to the four arrows shown on the keyboard keys **2**, **4**, **6**, or **8**.

FEED RATE — The **FEED** sets the speed at which the tool moves. The FEED rate is in inches per revolution (IPR).

NOTE: The tool will move as soon as the **START** key is pressed. To stop the tool movement, press the **ESC** key.

When the data values are entered, the screen reads:

```
>> CHECK LOCN
0000 PWRFD D04 0.0100
```

When the **START** key is pressed the tool will move (in this case to the left, at .01 i.p.r.).

8 ROUGH - This command is used to rough a piece of stock before a part is cut. The roughing cycle is a single pass box cut that is programmed by the end diameter and the ending Z axis coordinate. The cut begins at the location of the tool when the command is selected. The tool will return to the position it started from when the cut is complete. This means that the tool must be positioned before selecting the **ROUGH** command.

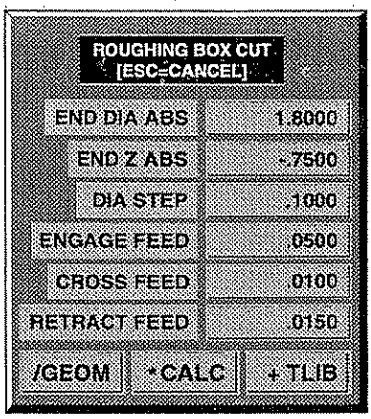


Figure 6-20

END DIA ABS This value sets the cutting depth of the roughing pass.

END Z ABS This value sets the endpoint of the roughing cut.

DIA STEP The **DIA STEP** is the value of the cutting depth that the machine will take, in the X axis, for each pass, toward the **END DIA ABS**.

ENGAGE FEED The **ENGAGE FEED** value is the speed at which the tool moves to the cut depth.

CROSS FEED The **CROSS FEED** is the speed at which the tool moves from the start point to the **END Z** point.

RETRACT FEED The **RETRACT FEED** is the speed at which the tool withdraws from the part. Remember that the tool is still in contact with the material at this point.

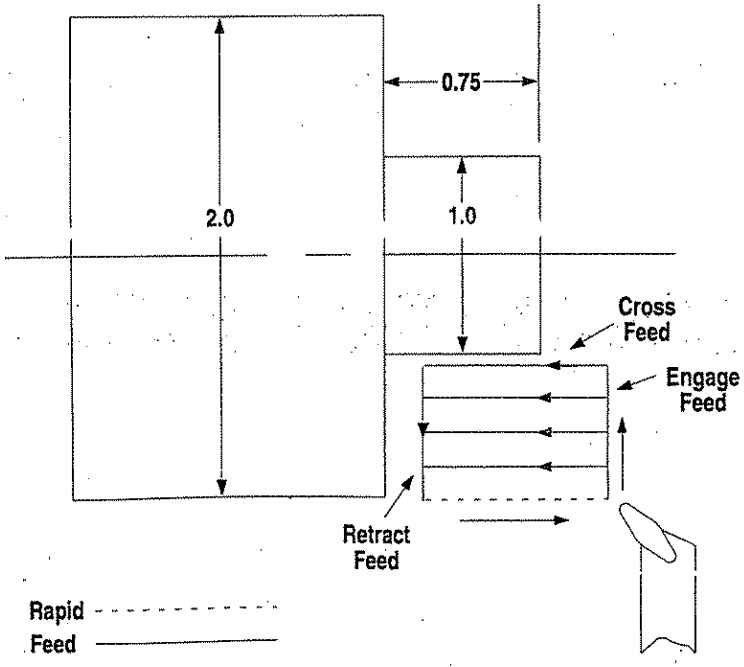


Figure 6-21

9 FACE - This command effects a FACING cut. The FACING cut is a box cut pattern parallel to the X axis. It is used to create a smooth face on the end of the workpiece.

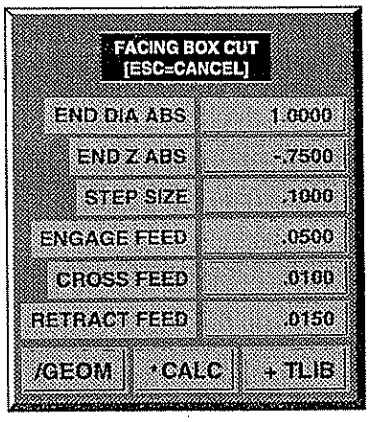


Figure 6-22

END DIA ABS The **END DIA** value sets the end point of the facing cut.

END Z ABS The **END Z** value sets the depth of the facing cut.

STEP SIZE The **STEP SIZE** value is the cutting step that the machine will take, in the Z axis, for each pass, toward the **END Z ABS**.

ENGAGE FEED The **ENGAGE FEED** value is the speed at which the tool moves to the cut depth.

CROSS FEED The **CROSS FEED** is the speed at which the tool moves from the start point to the **END DIA** point.

RETRACT FEED The **RETRACT FEED** is the speed at which the tool withdraws from the part. Remember that the tool is still in contact with the material at this point.

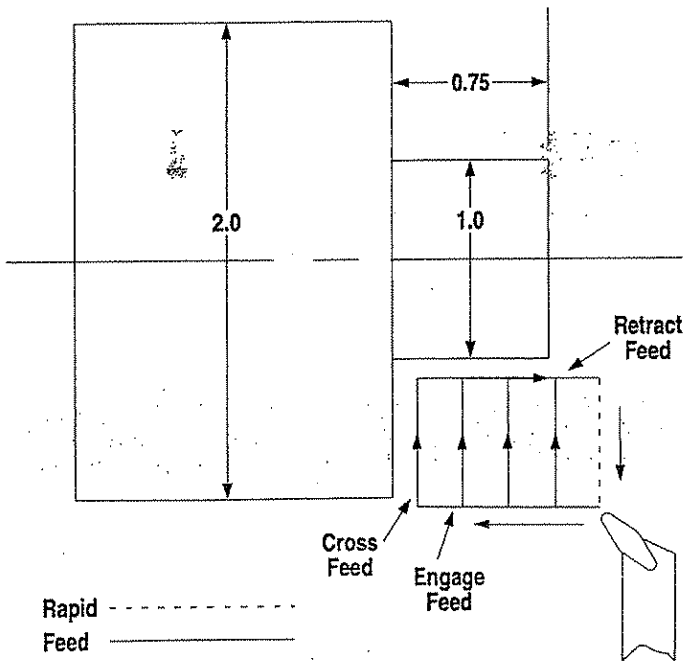


Figure 6-23

CHAPTER 7
MDI PROGRAMMING MODE

MDI PROGRAMMING

The MDI programming mode is used to manually input a programmed instruction sequence that is longer than one line. The MDI (manual data input) mode includes many of the instructions that are found in the DO EVENT mode, plus several features for more complex programming. When the MDI mode is selected by pressing 8 MDI from the BASIC OPERATIONS screen, the screen displays:

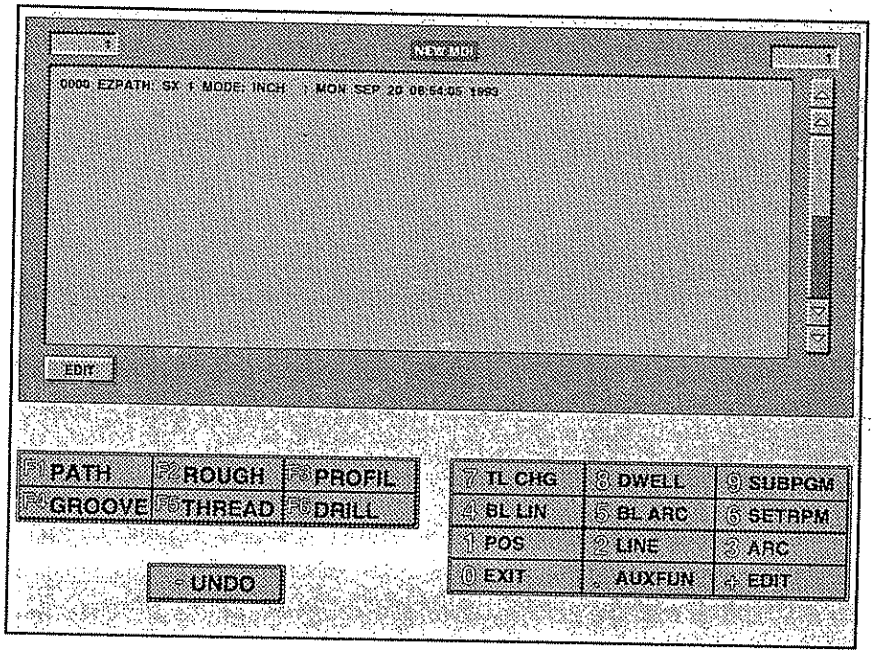


Figure 7-1

0 EXIT. The **0** key in the MDI mode exits the MDI screen and prompts the user to save the program. The program can be named at this point so that a new file is created. The program name can be typed in the **PRGM** field at this point. The program name can be up eight (8) numeric characters long. The file extension **.PGM** is added automatically.

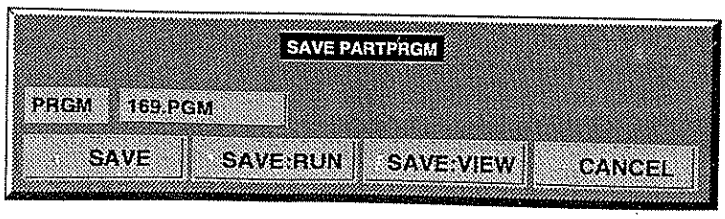


Figure 7-2

+ **SAVE**. This key causes the program to be saved under the name listed in the **PRGM** field shown above.

* **SAVE | RUN**. This command saves the program under the name in the **PRGM** field shown above, and loads the program into memory so that it can be executed.

/ **SAVE | VIEW**. This command saves the program under the name in the **PRGM** field shown above, and loads the program into memory so that it can be executed in the PREVIEW mode. For more information on PREVIEW, see chapter 12.

– **CANCEL**. This command cancels the **EXIT** command and returns to the EDITOR without saving the part program.

ESC(EXIT). This command exits the MDI mode without saving the part program.

. AUXFUN The **.** key (decimal point) can be used to enter several different commands into the part program.

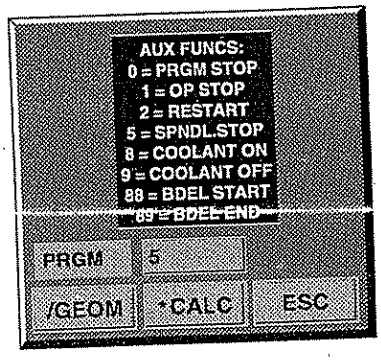


Figure 7-3

PRGM Enter the number of the command desired in this field. The available commands are listed at the top of the AUX FUNCS box. These commands are shown in the program with the code number. For instance, the PRGM STOP command is shown as:

```
0210 AUXFUN M0
```

The AUXFUN commands are explained below.

PRGM STOP The **PRGM STOP** command is used to halt the execution of a program. The carriage stops, and no action takes place until the **START** key is pressed by the operator. This is usually used at the end of a program, but may be used elsewhere also.

OP STOP The **OP STOP** command is similar to the **PRGM STOP** command, except that it is only executed if the **M1 OP STOP** option is enabled in the **/OPTS** menu in the run mode. If this option is not enabled, the **OP STOP** command is ignored. The spindle is **not** stopped automatically when an **OP STOP** is executed.

RESTART The **RESTART** command is used to reset the program back to the beginning. The program runs from the first line without stopping, unless a STOP command is programmed at the beginning of the program.

SPNDL STOP The **SPNDL STOP** command prompts the operator to stop the spindle rotation. The program execution also stops. To restart the program, start the spindle, then press the **START** key.

COOLANT ON The **COOLANT ON** command is used to enable the flood coolant when the spindle is turned on.

COOLANT OFF The **COOLANT OFF** command is used to disable the flood coolant.

BDEL START The **BDEL START** command signals the first line of a set of program instructions which may, or may not be executed. The **BLOCK DELETE** feature is enabled/disabled in the **RUN** mode when the program is executed.

BDEL END The **BDEL END** command signals the last instruction line that is omitted if the **BLOCK DELETE** option is **ENABLED**. All of the program lines between the **BDEL START** command and the **BDEL END** command are omitted.

+ EDIT The + key is used to transfer from the **MDI** mode to the **EDIT** mode. The part program currently in memory is not erased from the screen, but it is not saved either. To be certain that you save the program, press the **EXIT** key then enter a program name and press the + key to save the program.

1 POS The **POS** (position) command places a **RAPID** move in the program. This move will position the tool to the point specified by the DIA value and Z coordinate.

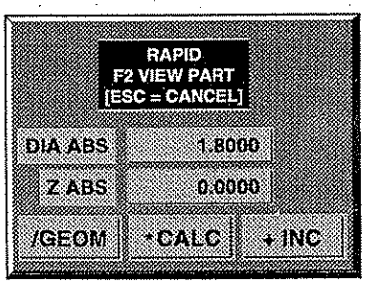


Figure 7-4

DIA ABS The **DIA** value entered here moves the tool carriage in the X axis. Note that the EZ-PATH X axis is programmed with part diameter values.

Z ABS The **Z ABS** value moves the tool carriage in the Z axis.

The **POS** command appears in the program as:

```
0010 RAPID ABS X1.8000 Z0.0000
```

This line produces the tool movement shown in Figure 7-5 below.

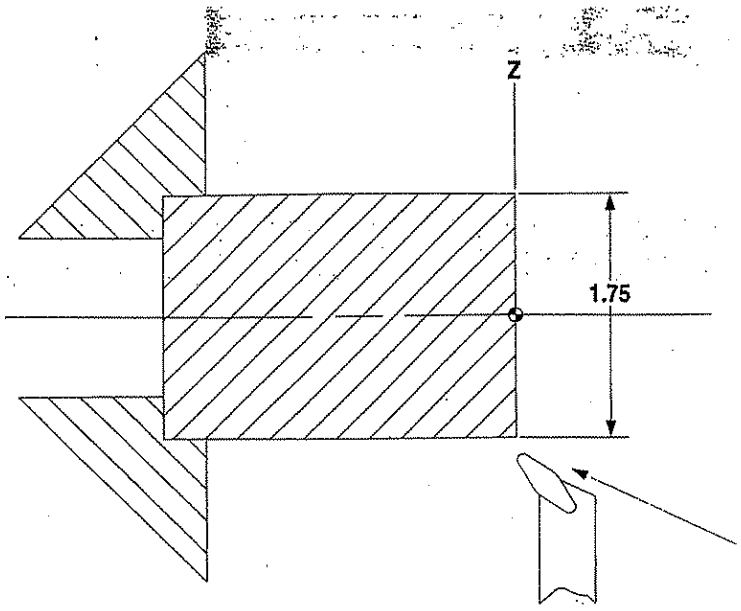


Figure 7-5

The **POS** command can also be programmed in the **incremental** mode, where the **DIA** and **Z** values are **distances** from the current tool location. To program the **POS** command in the **incremental** mode, press the **INC** key before entering any data.

2 LINE Pressing the **2** key places a **LINE** command in the part program. The **LINE** command causes the EZ-PATH to cut a straight line, from the current tool location to the location specified by the DIA value and Z coordinate. The tool moves at the programmed feedrate to the new location.

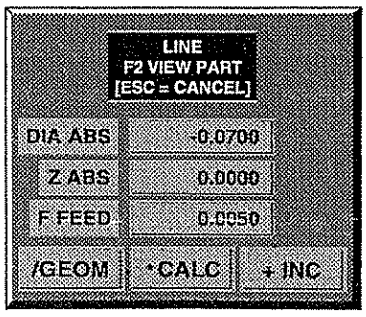


Figure 7-6

DIA ABS The **DIA** value entered here is the diameter that the tool is moving towards.

Z ABS The **Z** value entered here is the point that the tool is moving towards.

F FEED The **FEED** value entered here sets the speed at which the tool is cutting as it moves towards the target point. **FEED** values are always entered in **inches per revolution (IPR)**.

The **LINE** command appears in the program as:

```
0020 LINE ABS X-0.0700 Z0.0000 F0.0050
```

This program line produces the tool motion illustrated in Figure 7-7 below.

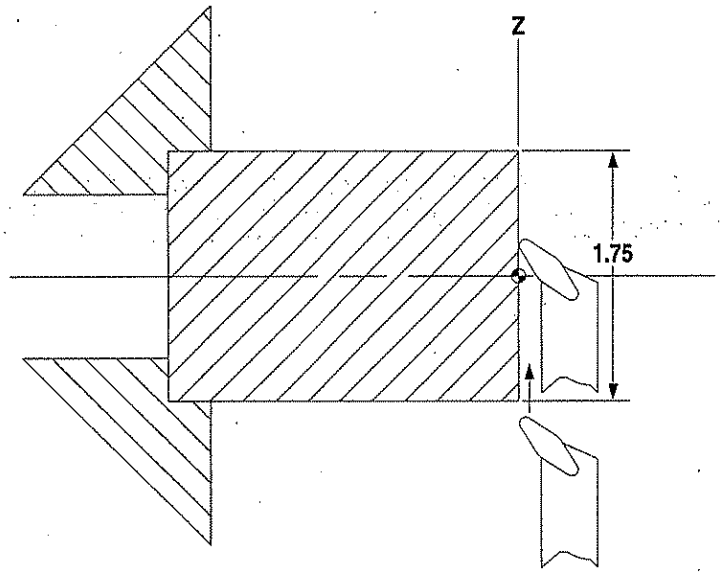


Figure 7-7

The **LINE** command can also be programmed in the **incremental** mode, where the **DIA** and **Z** values are **distances** from the current tool location. To program the **LINE** command in the **incremental** mode, press the **INC** key before entering any data.

3 ARC Pressing the **3** key places an **ARC** command in the part program. The **ARC** command causes the EZ-PATH to cut an arc in the chosen direction to the point specified by its DIA value and Z coordinate. The arc can be entered either with its centerpoint or with its radius.

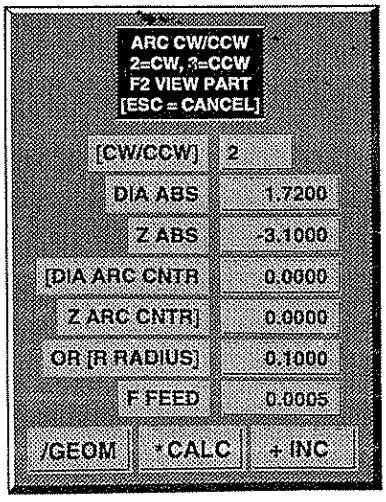


Figure 7-8

CW/CCW The value entered in the **CW/CCW** field determines the direction of the arc. Enter a **2** to cut the arc **clockwise**, or enter a **3** to cut the arc **counter-clockwise**.

DIA ABS This is the diameter of the arc endpoint.

Z ABS This value is the Z coordinate of the endpoint of the arc.

[DIA ARC CNTR
Z ARC CNTR]
OR [R RADIUS] The arc must be entered either with its centerpoint diameter and Z coordinate, or with its radius value. It is important not to enter values for all three fields here. Enter either the **DIA ARC CNTR** and **Z ARC CNTR** values, or the **R RADIUS** of the arc.

F FEED This is the cutting speed of the tool in inches per revolution (IPR).

The **ARC** command appears in a program as:

```
0030 ARC|RADIUS CW X1.7200 Z-3.1000 R0.1000 F0.0005
```

This program instruction produces the tool motion illustrated in Figure 7-9 provided the tool is positioned at X1.70 Z-3.0.

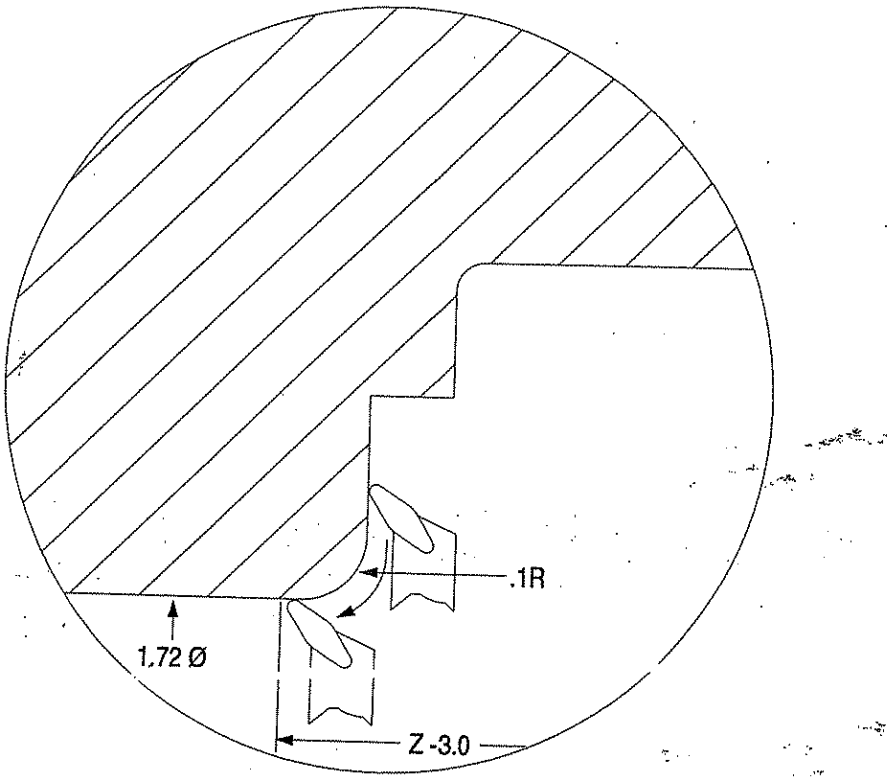


Figure 7-9

The **ARC** command can also be programmed in the **incremental** mode, where the **DIA** and **Z** values are **distances** from the current tool location. To program the **ARC** command in the **incremental** mode, press the **INC** key before entering any data.

4 BL LIN The 4 key in the MDI mode enters a **BL LIN** command in the part program. The **blend line** command cuts a straight line with a tangent arc at the end. The arc is also tangent to the line or arc programmed in the next command of the program. This command can also be used to cut a **chamfer** in place of the fillet arc.

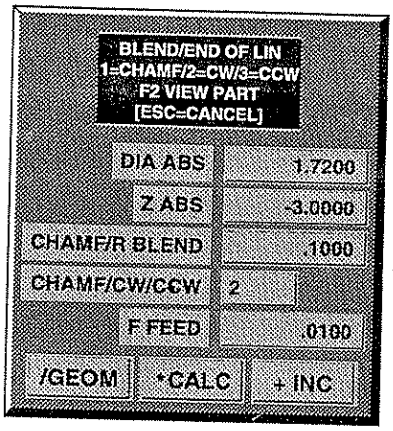


Figure 7-10

DIA ABS This is the diameter of the endpoint of the arc.

Z ABS This value is the Z coordinate of the endpoint of the arc.

CHAMF/R BLEND This value determines the fillet arc radius, or the chamfer distance from the corner point.

CHAMF/CW/CCW This value is either **1** for a chamfer, **2** for a **CW** fillet arc, or **3** for a **CCW** arc.

F FEED The **FEED** value sets the cutting rate in inches per revolution.

The **BL LIN** command appears in a program as:

```
0040 BLEND|LN ABS X1.7200 Z-3.0000 R0.1000 CW F0.0100
```

This program instruction produces the tool motion illustrated in Figure 7-11 provided the tool is positioned at X1.70 Z-3.0.

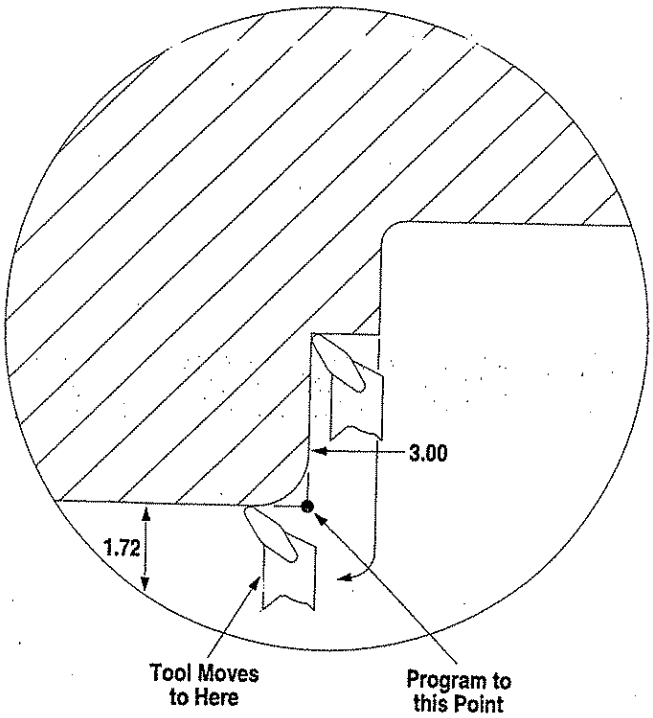


Figure 7-11

The **BL LIN** command can also be programmed in the **incremental** mode, where the **DIA** and **Z** values are **distances** from the current tool location. To program the **BL LIN** command in the **incremental** mode, press the + **INC** key before entering any data.

5 BL ARC The **5** key in the MDI mode enters a **BL ARC** command in the part program. The **blend arc** command cuts an arc with a tangent arc at the end. The arc is also tangent to the line or arc programmed in the next command of the program.

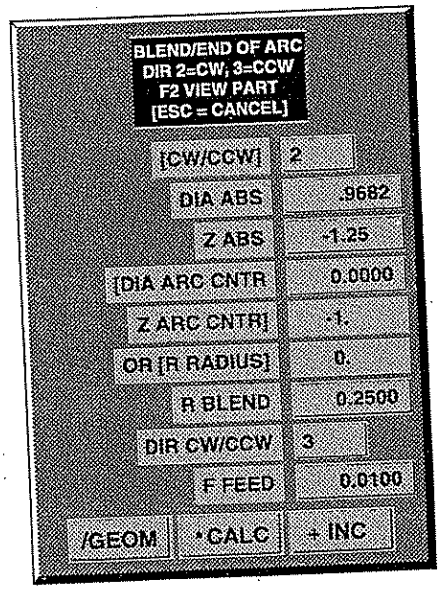


Figure 7-12

CW/CCW The direction of the arc is set with this value. Enter a **2** for a **clockwise** arc or a **3** for a **counter-clockwise** arc.

DIA ABS This value is the diameter of the end point of the arc.

Z ABS This value is the Z coordinate of the end point of the arc.

[DIA ARC CNTR
Z ARC CNTR]
OR [R RADIUS] The arc must be entered either by the diameter value and Z coordinate of its center, or by its radius. It is not necessary to fill in all three of these values.

R BLEND The **R BLEND** values is the radius of the fillet arc.

DIR CW/CCW This value sets the direction of the fillet arc.

F FEED This value sets the cutting speed of the tool, in inches per revolution.

The **BL ARC** command appears in a program as:

```
0050 BLEND|ARC|CNTRPT ABS CW X0.9682 Z-1.2500 XC0.0000 ZC-1.0000 R0.2500 CCW
```

This instruction produces the tool movement illustrated in Figure 7-13 provided the tool is positioned at X0.96 Z-1.25.

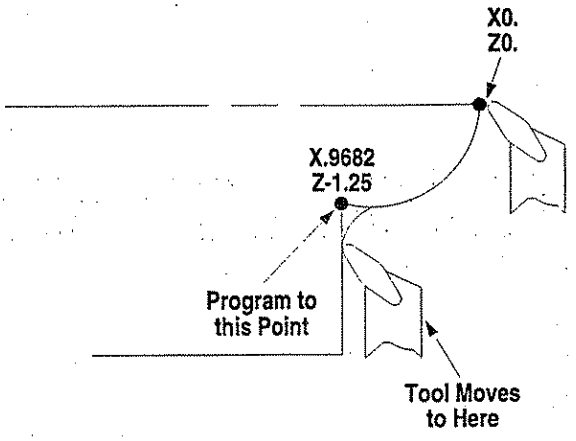


Figure 7-13

The **BL ARC** command can also be programmed in the **incremental** mode, where the **DIA** and **Z** values are **distances** from the current tool location. To program the **BL ARC** command in the **incremental** mode, press the **+ INC** key before entering any data.

6 SET RPM Pressing the **6** key enters a **SET RPM** command into the part program. This command does not set the spindle speed, instead it prompts the user to set the spindle speed to the programmed value.

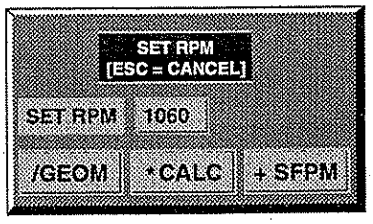


Figure 7-14

SET RPM This value sets a spindle speed.

NOTE: This command does not actually change the spindle rotation, instead it prompts the user to set the spindle rotation to the programmed speed.

The **SET RPM** command appears in a program as:

```
0010 SETRPM 1060
```

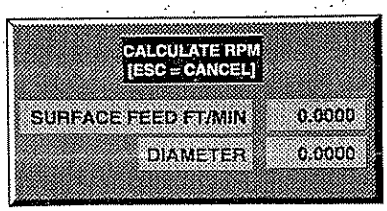


Figure 7-15

The **SET RPM** command has a Surface Feet Per Minute calculator that is accessed by pressing the + key from the **SET RPM** screen. This feature is used to calculate the proper spindle speed when the desired surface feed and the part diameter are known.

SURFACE FEED FT/MIN This is the value of the desired surface feed in feet per minute.

DIAMETER This is the diameter of the work piece.

7 TL CHG A tool change can be programmed by pressing the 7 key to enter a **TL CHG** command into the program. The tool ID must already be entered into the tool library before selecting this command, otherwise an error message is displayed.

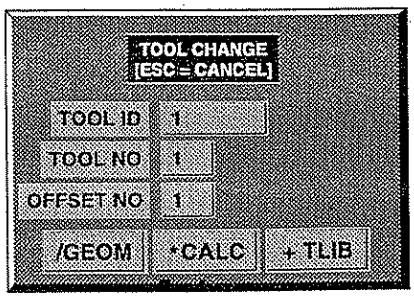


Figure 7-16

TOOL ID This is the Tool ID as it is entered in the Tool Library. A tool must be entered in the Tool Library before it can be used in a part program.

TOOL NO The **tool number** (1-24) refers to the tool position in the tool rack or tool holder. The operator is prompted for this tool number.

OFFSET NO This is the programmed **OFFSET** value (1-24) that should be used for the selected tool.

The **TL CHG** command appears in a program as:

```
0020 TLCHG I1 T01 01
```

8 DWELL The **8** key programs a **DWELL** time. This is used to keep the tool in place for the entered length of time. This may be used when programming a grooving cut using the LINE command.

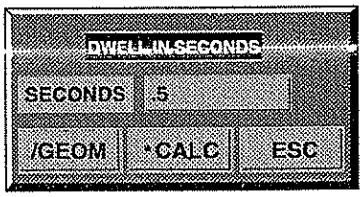


Figure 7-17

SECONDS This value determines how long the tool is paused in the current location.

The **DWELL** command appears in a program as:

```
0030 DWELL S0.5000
```

9 SUBPGM A part program may include a command which tells the EZ-PATH to execute another program. This second program is called a **sub-program**. One or more subprograms may be called by a part program, and each may be called more than once.

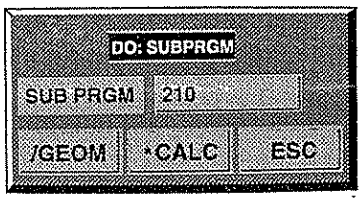


Figure 7-18

SUB PRGM The number entered here is the name of the **sub-program** as it is saved on the EZ-PATH.

The **SUB PRGM** command appears in a program as:

```
0050 DO|SUBPRGM[PGM] 210
```

This command would cause the EZ-PATH to execute the instructions contained in the PGM file named 210.PGM.

F1 PATH The **PATH** command allows the part contour (or shape) to be programmed once and then used in several different operations. For instance, in roughing and profiling a part the same path might be used. This lets the operator program the **PATH** once, then use only two more instruction lines to execute a **roughing** operation, and a **profiling** operation for the part.

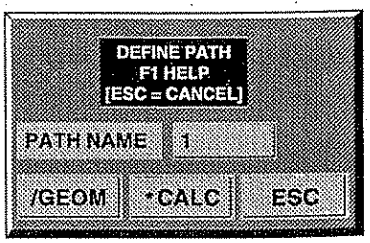


Figure 7-19

PATH NAME A **PATH** must have a unique name within a part program so that it is not confused with any other **PATH** in that program. The PATH name is a number, and can be up to six digits long. This name is used in commands which cut the PATH shape, such as roughing.

NOTE: The PATH must start with a positioning move to the beginning of the part.

For example:

```
0050 STARTPATH 1
0060 RAPID ABS X0.0000 Z0.0100
0070 LINE ABS X0.0000 Z0.0000 F0.0100
0080 LINE ABS X0.3000 Z0.0000 F0.0100
0090 LINE ABS X0.3000 Z-0.1000 F0.0100
0100 LINE ABS X0.4000 Z-0.1000 F0.0100
0110 LINE ABS X0.4900 Z-0.1500 F0.0100
0120 LINE ABS X0.4900 Z-0.6250 F0.0100
0130 LINE ABS X0.7500 Z-1.6250 F0.0100
0140 LINE ABS X0.7500 Z-1.7500 F0.0100
0150 LINE ABS X0.9000 Z-1.7500 F0.0100
0160 LINE ABS X1.0000 Z-1.8000 F0.0100
0170 LINE ABS X1.0000 Z-2.8750 F0.0100
0180 BLEND|LN ABS X1.3750 Z-2.8750 R0.0100 CW F0.0100
0190 LINE ABS X1.3750 Z-3.0000 F0.0100
0200 BLEND|LN ABS X1.7200 Z-3.0000 R0.1000 CW F0.0100
0210 LINE ABS X1.7200 Z-3.7500 F0.0100
0220 LINE ABS X1.9000 Z-3.7500 F0.0100
0230 PATHSTOP
0240 ROUGH 1 I1 X0.0200 Z0.0050 F0.0100 0.0100 0.0100 S0.1000
C0.1000W45.0000W0.0500D2U1A1
0250 RAPID ABS X1.0000 Z0.5000
0260 PROFIL 1 1 X0.0000 Z0.0000 F0.0100 C0.1000 E45.0000 W135.0000 U1 A1
```

In the program above, lines 50 – 230 establish the **PATH** shape. The shape for the part shown in Figure 7-20 below.

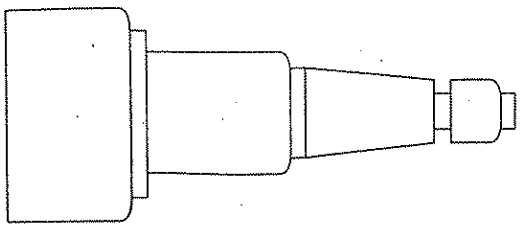


Figure 7-20

The machines does not actually cut this shape until line 240, the **ROUGH** command which instructs the machine to rough out the shape that is defined as **PATH 1**.

Line 250 repositions the tool, and line 260 instructs the machine to **PROFILE** the shape defined in **PATH 1**.

NOTE: **PATH** is used to define three things about a part

1) It defines the part shape
2) It defines the direction the tool moves in **PROFILE**
3) It defines the stock size or material to be removed, by the intersection of the first position in the **PATH** and the last position in the **PATH** (as shown in Figure 7-21 below)

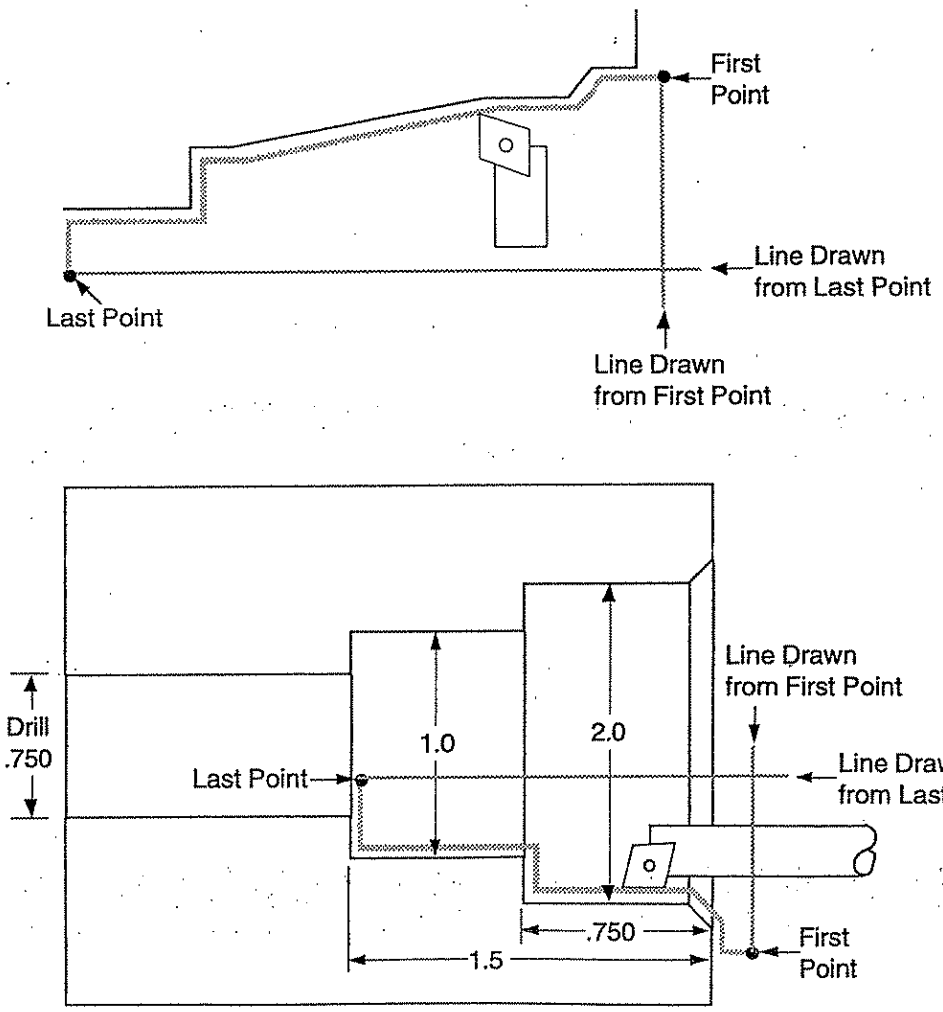


Figure 7-21

The following example shows the program lines for a boring operation as shown in Figure 7-21 above:

```
0020  STARTPATH 1
0030  RAPID ABS X2.2000 Z0.0500
0040  LINE ABS X2.2000 Z0.000 F0.0050
```

```
0050   CHAMFER ABS X2.0000 Z0.0000 F0.0050
0060   LINE ABS X2.0000 Z0.0000 P0.0500 P0.0500 F0.0050
0070   LINE ABS X1.0000 Z-0.7500 F0.0050
0080   LINE ABS X1.0000 Z-1.5000 F0.0050
0090   LINE ABS X0.7500 Z-1.5000 F0.0050
0100   PATHSTOP
0110   ROUGH 1 I2 X0.0000 Z0.0000 F0.0100 0.0100 S0.1000 C0.1000 W45.0000
       W0.0500 D2 U1 A1
```

In the program above, lines 20 – 100 define the **PATH** or **PART** shape. The part is cut in line 110 with the **ROUGH** command.

The **F1** key indicates the beginning of a **PATH** with a **STARTPATH** statement in the program. The **PATH** itself can be defined with only **POS**, **LINE**, **ARC**, **BL LIN**, and **BL ARC** commands. The end of the **PATH** is indicated by a **PATHSTOP** command.

When the **PATHSTART** statement is entered into the part program, the key functions are limited to the five commands allowed to define a path, and the **PATHSTOP** command. Once **PATHSTOP** is selected, the key functions return to the normal MDI mode.

> **NOTE:** all feed rate values in the path are ignored since the actual feed rates are set by the rough, profile or groove cycles.

F2 ROUGH The **ROUGH** command is used to program a roughing operation for a part shape that is defined in a **PATH**. The **ROUGH** command allows the operator to enter finish allowances, engage, cross and retract feedrates, the cut step, clearance, withdraw angle and length, and to turn off undercut checking and auto-rounding if desired.

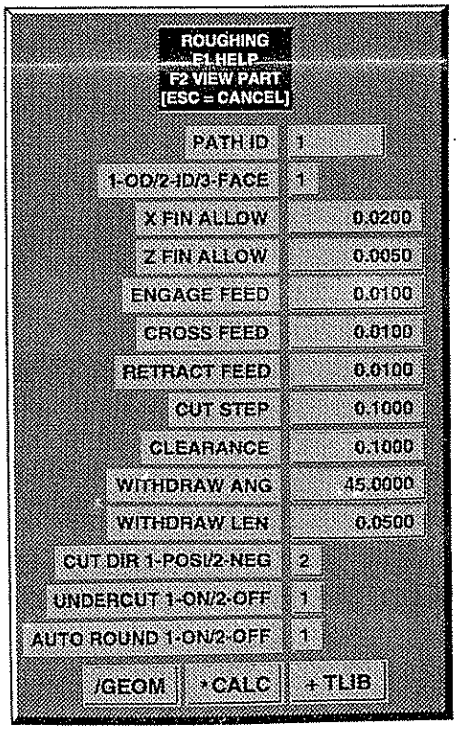


Figure 7-22

PATH ID — The **PATH ID** must be the name of a **PATH** that has been defined previously in the program.

1-OD/2-ID/3-FACE — This value determines where the roughing operation takes place. This may be an OD (outer diameter) operation, an ID (inner diameter) operation, or a FACE (part end) operation.

X FIN ALLOW — The **X FIN ALLOW** is the amount of material left on the diameter of the part, that is to be removed in a finishing operation.

Z FIN ALLOW — The **Z FIN ALLOW** is the amount of material left on the vertical faces of the part, that is to be removed in a finishing operation

ENGAGE FEED The **ENGAGE FEED** determines the feedrate of the tool as it plunges toward the part center line. Feed rates are in inches per revolution.

CROSS FEED The **CROSS FEED** sets the feedrate of the tool as it cuts in the X-axis only. Feed rates are in inches per revolution.

RETRACT FEED The **RETRACT FEED** sets the feedrate of the tool as it withdraws from the part center line. Feed rates are in inches per revolution.

CUT STEP The **CUT STEP** parameter determines the depth of each cutting pass of the tool. The tool will make several cutting passes to remove all of the material down to the **PATH** shape (plus the finish allowance).

CLEARANCE The **CLEARANCE** value sets the distance the tool retracts from the part before the engage feed move for the next cutting pass.

WITHDRAW ANG The **WITHDRAW ANGLE** is the angle at which the tool is retracted from the part at the end of each cutting pass. See Figures 7-23 and 7-24.

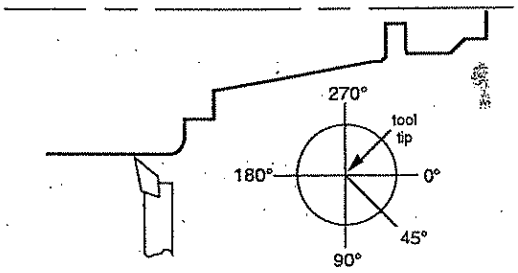


Figure 7-23

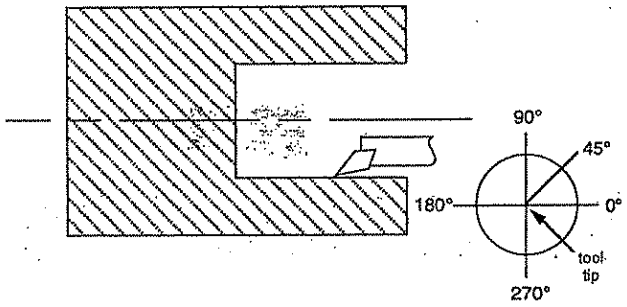


Figure 7-24

WITHDRAW LEN The **WITHDRAW LENGTH** value is the distance that the tool moves to disengage from the part at the end of each cutting pass.

CUT DIR 1-POS/2-NEG The **CUT DIRECTION** is determined by the tool definition, but it may be either **NEGATIVE** (towards the headstock) or positive (towards the tailstock).

UNDERCUT 1-ON/2-OFF The EZ-PATH uses an **UNDERCUT** checking feature in its software. This feature may be disabled by entering a **2** here. With this feature is turned on, a note will be displayed during program load if undercut is detected. Press enter to continue.

AUTO ROUND 1-ON/2-OFF The EZ-PATH uses an **AUTO ROUND** feature which places a small arc, equal to the tool radius at each corner. This still leaves a sharp corner, but it decreases the tool movement. In some cases it may be necessary to turn this feature off. Enter a **2** here to turn off the **AUTO ROUND** feature.

The **ROUGH** command appears in a program as:

```
0240 ROUGH 1 I1 X0.0200 Z0.0050 F0.0100 0.0100 0.0100 S0.1000
C0.1000W45.0000W0.0500D2U1A1
```

This command roughs the shape for the part shown in Figure 7-20. The "path" program is shown in the **PATH** command description.

F3 PROFIL The **PROFIL** command is used to program a profiling operation for a part shape that is defined in a **PATH**. The **PROFIL** command allows the operator to enter finish allowances, a feedrate, clearance, and withdraw angle, and to turn off undercut checking and auto-rounding if desired.

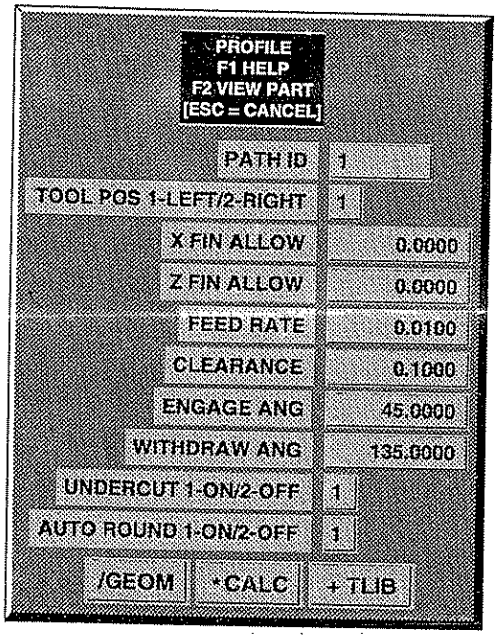


Figure 7-25

PATH ID The **PATH ID** must be the name of a **PATH** that has been defined previously in the program.

TOOL POS 1-LEFT/2-RIGHT This value determines where the tool is placed in relation to the PATH. The tool position is determined by looking along the path in the cutting direction and setting the tool on the correct cutting side (left or right).

X FIN ALLOW The **X FIN ALLOW** is the amount of material left on the diameter of the part (after the roughing pass), and to be removed in a finishing operation.

Z FIN ALLOW The **Z FIN ALLOW** is the amount of material left on the vertical faces of the part, and to be removed in a finishing operation.

FEED RATE The **FEED RATE** entered here is used all through the **PROFILE** operation.

CLEARANCE The **CLEARANCE** value sets the distance through which the tool moves to engage the part, and to disengage from the part.

ENGAGE ANG The **ENGAGE ANGLE** sets the angle at which the tool moves to engage the part.

WITHDRAW ANG The **WITHDRAW ANGLE** is the angle at which the tool is retracted from the part at the end of each cutting pass. The Withdraw Angle in Profile differs from Withdraw Angle in Rough in that the angle is determined by the direction of the last move. If the last move is an X positive move, for OD turning, the Withdraw Angle will be as shown in Figure 7-26.

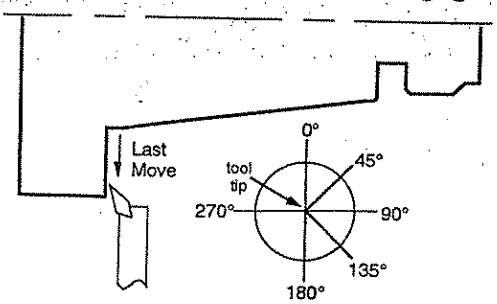


Figure 7-26

UNDERCUT 1-ON/2-OFF The EZ-PATH uses an **UNDERCUT** checking feature in its software. This feature may be disabled by entering a **2** here. Usually, it is better to leave this feature turned on.

AUTO ROUND 1-ON/2-OFF The EZ-PATH uses an **AUTO ROUND** feature which places a small arc, equal to the tool radius, at each corner. This still leaves a sharp corner, but it decreases the tool movement. In some cases it may be necessary to turn this feature off. Enter a **2** here to turn off the **AUTO ROUND** feature.

The **PROFILE** command appears in a program as:

```
0260 PROFIL 1 1 X0.0000 Z0.0000 F0.0100 C0.1000 E45.0000 W135.0000 U1 A1
```

This command is the **PROFILE** command for the part shape shown in Figure 7-20. The "path" program is shown in the **PATH** command description.

F4 GROOVE The **GROOVE** command programs the EZ-PATH to cut a groove in the shape given in a **PATH**. The **GROOVE** command allows the operator to enter a finish allowance, plunge and retract feeds, clearance, step over, lift off, and dwell values.

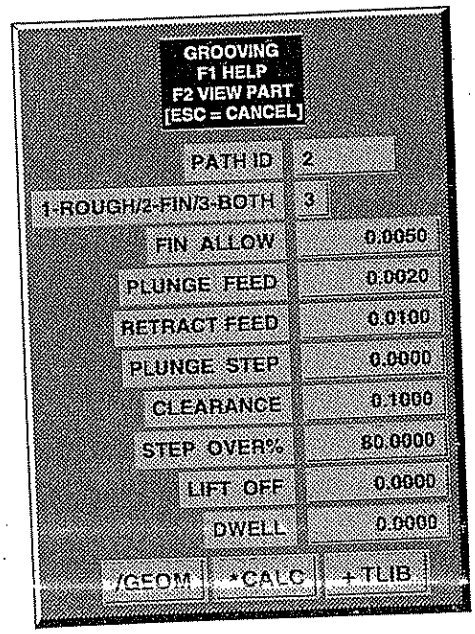


Figure 7-27

PATH ID The **PATH ID** must be the name (number) of a **PATH** that has been defined previously in the program.

1-ROUGH/2-FIN/3-BOTH The **GROOVING** command can be used to **rough** or **finish** or do **both** operations. Enter a **1** here to **rough**, a **2** to **finish**, or a **3** to do **both**.

FIN ALLOW The **FINISH ALLOWANCE** value sets the amount of material that remains on the part (after the roughing pass) and will be removed by the **finish** pass.

PLUNGE FEED The **PLUNGE FEED** value sets the rate at which the tool feeds into the part (in inches per revolution).

RETRACT FEED The **RETRACT FEED** value sets the speed at which the tool is removed from the part.

PLUNGE STEP The **PLUNGE STEP** value sets the depth of each of the plunge moves.

CLEARANCE The **CLEARANCE** value sets how far from the part the tool begins its plunge move. The **CLEARANCE** value is measured from the highest point of the groove PATH shape.

STEP OVER% The **STEP OVER%** value sets the percent of the tool width that the tool is moved between plunge moves. For example, if the tool has a width of 0.3, and the STEP OVER% is 50, then the tool is moved 0.15 between each plunge move.

LIFT OFF The **LIFT OFF** value sets the distance that the tool moves at the end of a plunge cut. This is a shift in the opposite direction of the cutting direction. The **LIFT OFF** distance moves the tool away from the uncut material before the tool is retracted from the part at the rapid feed rate. The **LIFT OFF** move is not made at the end of the first plunge move.

DWELL The tool can be set to remain in position after each plunge move for a period of time. The **DWELL** value sets this time in seconds.

A **GROOVE** path would appear as:

```
0190 STARTPATH 2
0200 RAPID ABS X1.0500 Z-1.2500
0210 LINE ABS X0.8000 Z-1.2500 F0.0100
0220 LINE ABS X0.8000 Z-0.8500 F0.0100
0230 LINE ABS X1.0500 Z-0.8500 F0.0100
0240 PATHSTOP
0250 GROOVE 2 3 A0.0050 F0.0020 R0.0100 P0.0000 C0.0250 O80.0000 L0.0000
D0.0000
```

In the example above, the PATH is defined in lines 0190 through 0240. The material is not cut however, until line 0250 which cuts the groove in the shape defined in the PATH. This example would be used to cut the groove shown below in Figure 7-28.

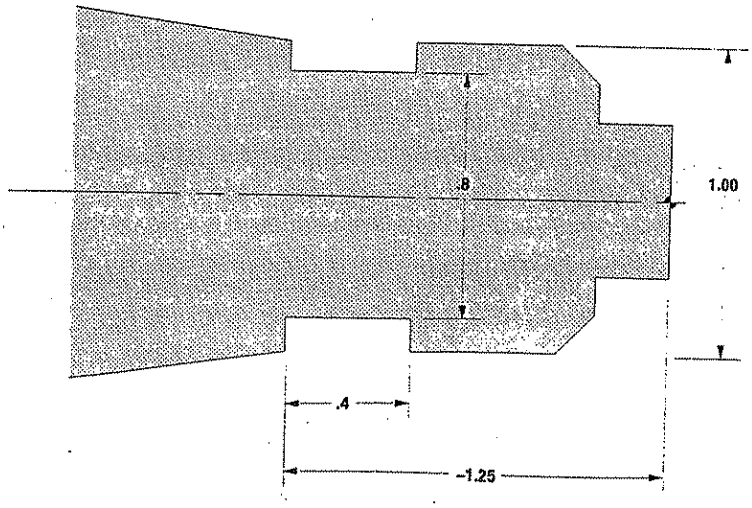


Figure 7-28

F5 THREAD The **THREAD** cycle is used to program the EZ-PATH to cut threads on a particular area of the part.

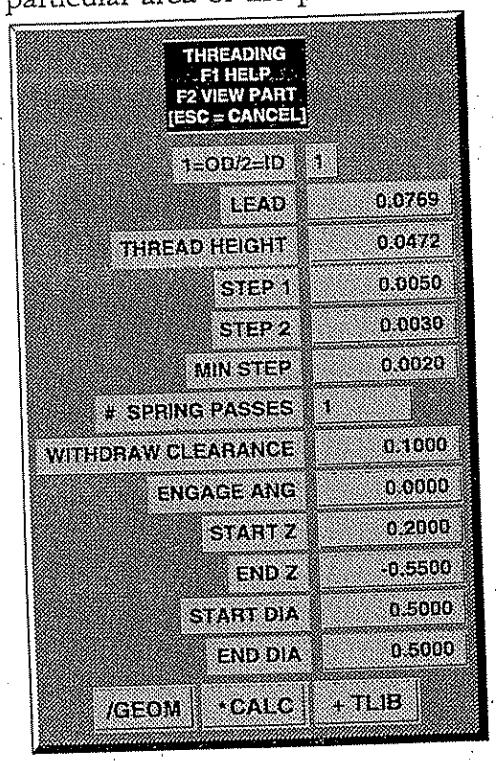


Figure 7-29

1=OD/2=ID Threads can be cut on either the inside diameter (**ID**) of the part or the outside diameter (**OD**) of the part. Enter a **1** here to cut **OD** threads, or a **2** to cut **ID** threads.

LEAD The **LEAD** value is the distance the screw advances axially in one turn. This can be calculated as **1/t.p.i.**. For example:

if there are 13 threads per inch,

then the **LEAD** is 1/13 t.p.i. (or 0.0769).

THREAD HEIGHT The thread height determines how deep the threads are cut. This is the distance from the top to the bottom of one thread. A standard thread height is automatically calculated for a 60° UN thread.

STEP 1 The **STEP 1** value sets the cutting depth of the first pass of the threading tool.

STEP 2 The **STEP 2** value sets the cutting depth for the second pass of the threading tool. All subsequent steps are equal to the value of STEP 2 until the amount of *material left on the part* equals the value of STEP 2. The machine then reduces the cut depth by half for each step until the MIN STEP value is reached.

MIN STEP The **MIN STEP** value sets the smallest cutting depth allowed.

 # SPRING PASSES Often, it is necessary to make additional passes without changing the cutting depth of the tool when cutting threads. Since the tool exerts a great deal of pressure on the part material, the part may be slightly deformed so that the threads are not cut accurately. Making additional **SPRING PASSES** which do not change cutting depth and do not exert great pressure on the part, makes certain that the threads are cut accurately.

WITHDRAW CLEARANCE The **WITHDRAW CLEARANCE** value sets the distance from the part that the tool is retracted before it rapid traverses to the start point for another threading pass. This distance is measured from the top of the threads.

ENGAGE ANGLE The angle of the tool as it feeds into the work piece. It is positive for both OD and ID threads.

START Z The **START Z** sets the location where the thread cutting pass starts. Usually this is not on the material but close to it.

END Z The **END Z** value sets the location where the thread cutting pass ends. Usually this is not on the part material, but close to it. Generally, undercuts are made to either side of the area where the threads will be cut. This is done before the threading operation so that the tool has sufficient room to clear the material before engaging or disengaging the part.

START DIA The **START DIA** value sets the nominal diameter of the thread.

END DIA The **END DIA** value sets the nominal diameter of the thread. The **END DIA** value is usually the same as the **START DIA** value. If the threads being cut are tapered threads or pipe threads, then this value is used by the EZ-PATH to calculate the angle of the threads.

The **THREAD** command appears in a part program as:

```
0110 THREAD 1 L0.0769 H0.0472 S0.0050 0.0030 0.0020 #1 C0.1000 Z0.2000 -
0.5500 D0.5000 0.5000
```

This **THREAD** command would produce the threads shown in Figure 7-30.

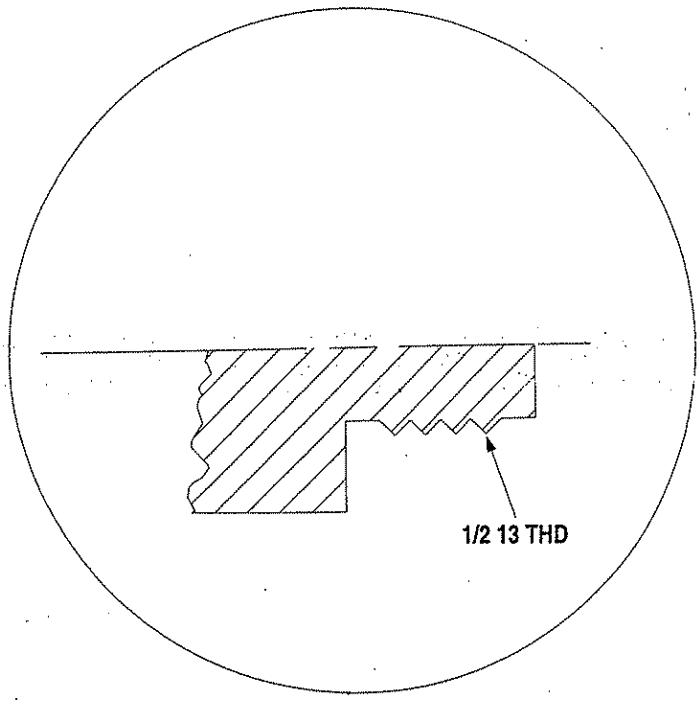


Figure 7-30

F6 DRILL The **DRILL** cycle is used to drill out the center of the part. The starting Z coordinate, depth, step values, clearance, and feedrate values must be entered.

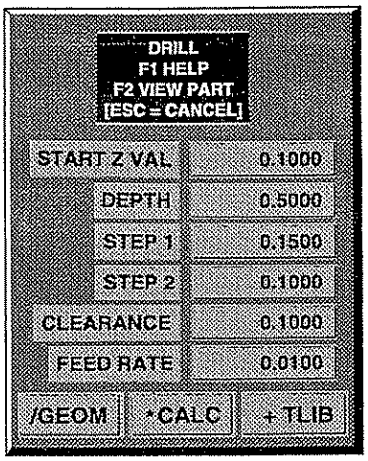


Figure 7-31

START Z VAL The **START Z** value sets the position at which the tool starts the drilling operation.

DEPTH The **DEPTH** value sets distance that the drill moves from the **START Z** position.

STEP 1 The **STEP 1** value sets the depth of the first drill plunge.

STEP 2 The **STEP 2** value sets the depth of each plunge after the first.

CLEARANCE The **CLEARANCE** value is the distance from the part that the drill is withdrawn when the drilling operation is finished.

FEED RATE The **FEED RATE** value sets the speed that the drill feeds into the part during the drilling operation.

A **DRILL** command appears in a part program as:

```
0300 DRILL Z0.1000 D0.5000 T0.1000 0.1500 C0.1000 F0.0100
```

This program instruction would produce the drill cycle illustrated in Figure 7-32.

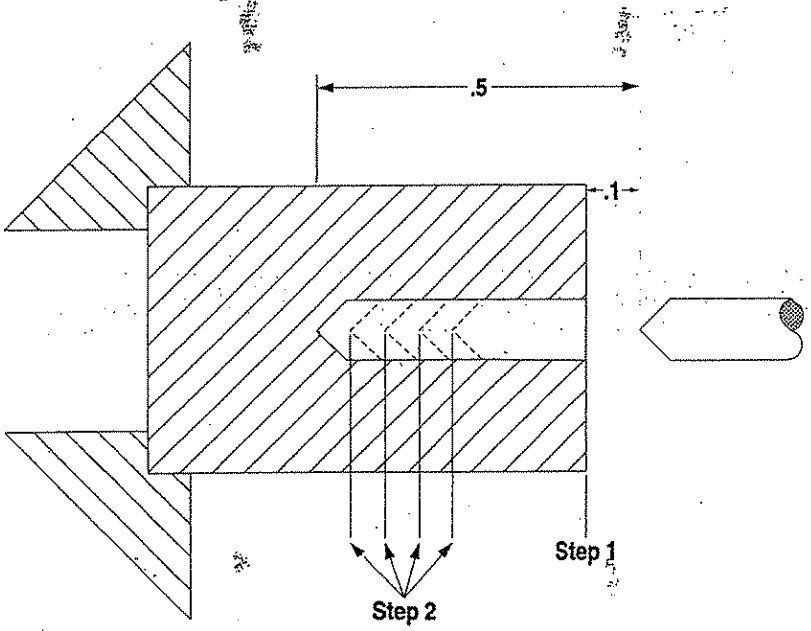


Figure 7-32

CHAPTER 8
EDIT MODE

The EZ-PATH has a dedicated Program Editor which is called from the BASIC OPERATIONS screen by pressing the **5** key. This is an intelligent editor and is capable of recognizing each command in the part program. When a line is selected for editing, the data is shown in the fields used to program the line originally.

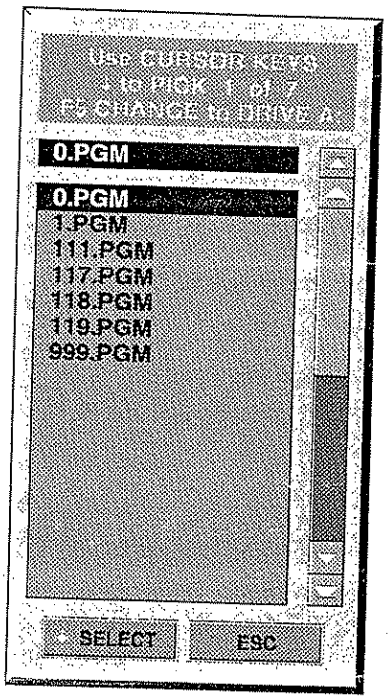


Figure 8-1

When the EDITOR is selected from the BASIC OPERATIONS screen a list of files that are currently available for editing is displayed, as shown in Figure 8-1.

A file is selected from this list by using the up and down cursor arrow keys or the page up/page down keys to move the highlight bar to select the desired file.

When the file to be edited is highlighted, press the + key to select the file and load it into the EDITOR.

Pressing the **ESC** key while the list of editable files is shown on the screen aborts the call to the EDITOR and returns the display to the BASIC OPERATIONS screen. Pressing **F5** selects the floppy drive.

The EDITOR also provides specific commands for inserting and deleting lines, renumbering the program lines, and setting the line numbers in the part program.

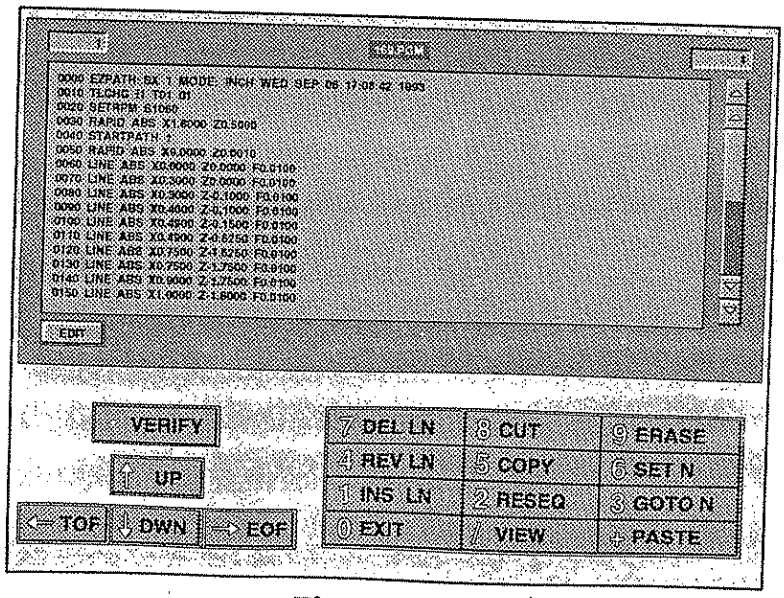


Figure 8-2

0 EXIT. The **0** key in the EDIT mode exits the EDITOR and prompts the user to save the edited program. The name can be changed at this point so that a new file is created, leaving the original file unchanged. The new program name can be typed in place of the existing name in the **PRGM** field. The program name can be up to eight (8) numeric characters. The file extension **.PGM** is added automatically.

+ SAVE. This key causes the program to be saved under the name listed in the **PRGM** field shown above. If this name is not changed, then the old file is replaced with the program just edited.

*** SAVE | RUN**. This command saves the program under the name in the **PRGM** field shown above and loads the program into memory so that it can be executed.

/SAVE | VIEW. This command saves the program under the name in the **PRGM** field shown above and immediately enters the **PREVIEW** screen.

– CANCEL. This command cancels the **EXIT** command and returns to the EDITOR without saving the part program.

ESC(EXIT). This command exits the EDIT mode without saving the part program. Any changes that were made during the EDIT mode session are not saved and the program remains in its original condition.

/ VIEW. The **VIEW** command displays the part program geometry on the screen. The operator is prompted to input the first and last line numbers to be viewed. When the part is viewed, a prompt appears to either **EXIT** the VIEW mode or **REVIEW**. The **REVIEW** command allows the user to enter new line numbers to view the part again.

*** VERIFY**. This command is used to check the actual movement of the tool when the program is run. It can only be used with canned cycles, such as ROUGH, PROFILE, GROOVE, THREAD, and DRILL. After the computer computes the necessary move a screen appears as shown in Figure 8-3. The dialogue box gives several options for viewing the part.

0 EXIT This command will return to the main editor.

1 RESIZE This command will cause the viewing window to resize so that a closer look can be taken at a section of the part.

2 STEP MODE This command is used to view the machining steps one at a time. The user can either press 0 to view steps singly or 1 to view all the steps in rapid succession.

3 SHOW TOOL This command will cause a representation of the tool tip to appear. The user can have the tool tip showing and use step mode to get a clear picture of what movements the tool will make.

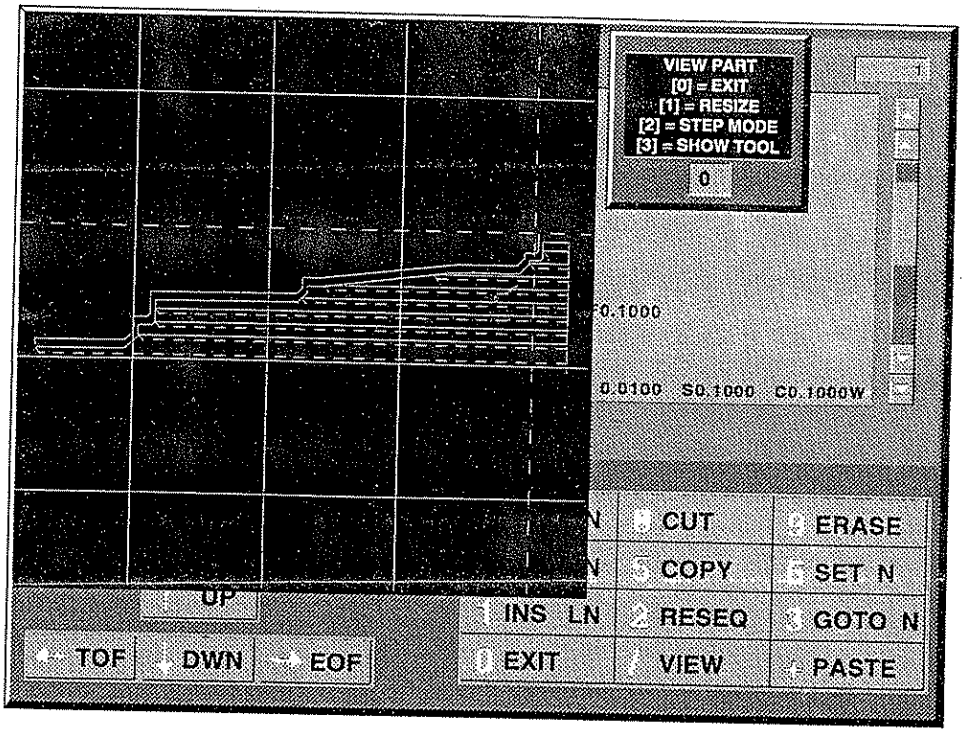


Figure 8-3

\+ **PASTE**. This command inserts the text from the temporary buffer COPY.TMP into the program **after** the current line. The buffer COPY.TMP is **not** destroyed after the text is pasted, so that the text may be pasted multiple times.

1 **INSERT LN**. Pressing the **1** key in the EDITOR calls the MDI programming screen so that lines can be inserted into the current program **after** the current line. Lines are inserted using the MDI mode by selecting the desired commands and entering the necessary data for each line. The MDI mode remains active until the **0** key (**XIT INS**) is pressed. This key returns to the EDITOR. After returning to the EDITOR from the MDI mode it is a good idea to **resequence** the part program by pressing the **2** **RESEQ** key.

2 **RESEQ**. The **2** key in the EDITOR performs a **resequence** command on the part program being edited. The screen displays prompts for the **N,SEQNO** number (the line number of the first program line) and **N,INC** (the increment for each of the following line numbers). The part program is automatically renumbered when these two numbers are entered. Pressing **ESC** aborts the resequence command and returns to the EDITOR without affecting the program.

3 **GOTO N**. This command searches the program for the next occurrence of a designated sequence number. For instance, if **150** is entered, the cursor moves to line 150. GOTO works from the current cursor position down to the end of the program. It will not search backwards.

4 **REV LN**. This command is used to revise any line in the part program. Using the cursor arrow keys, place the cursor on the line to be revised. Then press the **4 (REV LN)** key. The line is read and broken down into the data fields that were used to create the operation. The data in each field of the command can be changed with a few exceptions. The nature of a command cannot be changed, for example, a line with the M ARC operation cannot be changed to M LINE. This must be changed by deleting the offending line and then inserting a new line.

NOTE: the operator cannot change the context of the block of data. If the block was created as a MILL ARC by CENTERPT command it cannot be modified to a MILL ARC by RADIUS command. To make a command type change, delete the line and insert a new one with the desired context. Also, note that the first line in the part program, **0000 EZPATH|SX 1 MODE|INCH**, cannot be edited.

5 **COPY**. The **COPY** command is used to copy a group of lines into a temporary buffer named COPY.TMP. The screen displays prompts for **BEGIN at SEQNO** (the number of the first line to be copied) and **END at SEQNO** (the line number after the last line to be copied). The line with the designated **END at SEQNO** will **not** be copied. **COPY** is use in conjunction with the **PASTE** command. **NOTE:** the buffer, COPY.TMP, will contain the last text copied so that it can be used as a "clipboard" to paste data from one program into another. The cursor must be above the first line to be copied.

6 **SET N**. This command is used to control the line numbers of any lines which are inserted into the part program by selecting the **1 INS LN** command. The screen displays prompts for **N,SEQNO** the first line number, and **N,INC** the increment of each following line number.

7 **DEL LN**. This command deletes the line on which the cursor currently appears. It is a good idea to use the **2 RESEQ** command after deleting one or more lines from a part program. DO NOT DELETE THE FIRST LINE OF THE PART PROGRAM.

8 **CUT**. The **CUT** command is similar to the **COPY** command. A group of lines is copied to the temporary buffer COPY.TMP; however, when the selected lines are copied, they are also deleted from their original location. This command is useful for moving a group of lines from one section of a program to another. The cursor must be above the first line to be cut.

9 **ERASE**. The **ERASE** command is used to delete a group of lines at one time. The screen displays prompts for **BEGIN at SEQNO** (the number of the first line to be deleted) and **END at SEQNO** (the number of the last line to be deleted). The line with the designated **END at SEQNO** will **not** be deleted. The cursor must be above the first line to be erased.

<right arrow> EOF. This key moves the cursor to the last line of the part program.

<left arrow> TOF. This key moves the cursor to the first line of the part program.

<up arrow> Up. This key moves the cursor upward from the current line to the previous line on the screen. This is useful for positioning the cursor before using the

PASTE command to insert text that has been cut or copied to the COPY.TMP buffer.

<down arrow>. This key moves the cursor downward from the current line to the following line on the screen. This is useful for positioning the cursor before using the **PASTE** command to insert text that has been cut or copied to the COPY.TMP buffer.

CHAPTER 9
RUN MODE

INTRODUCTION

The **RUN** mode is used to execute a pre-loaded program. The program can be executed in **BLOCK** mode one instruction at a time with machine stops between each instruction, or it can be executed in **AUTO** mode with continuous operation. The **RUN** command is chosen from the BASIC OPERATIONS screen by pressing the + key. When the **RUN** command is chosen, the screen displays:

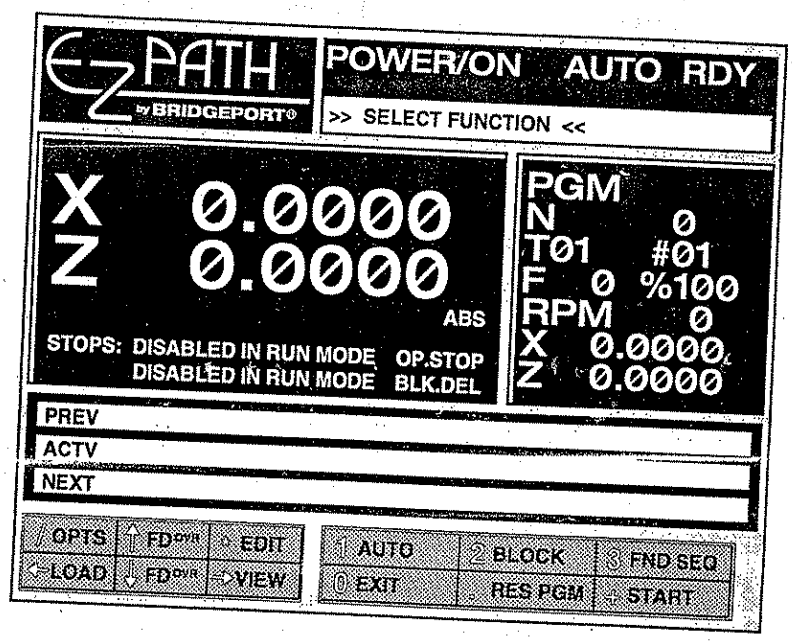


Figure 9-1

KEY FUNCTIONS

0 EXIT The **EXIT** command leaves the **RUN** mode and returns to the BASIC OPERATIONS screen.

1 AUTO The **AUTO** command sets the loaded program to be run in continuous operation. The program stops for spindle speed changes, tool changes or programmed stops. The program run begins when the **START** key is pressed.

2 BLOCK The **BLOCK** commands sets the loaded program to be run in single step mode. Each line of a program is executed and the program is halted until the **START** key is pressed. The program run begins when the **START** key is pressed.

3 **FND SEQ** This command searches the part program for the designated sequence number. When the START command is used, part program execution will begin at the entered sequence number. FIND SEQUENCE only searches down through the program. It may be necessary to resequence the program before using this command.

. **RES PGM** This command is used to reset the active program back to the beginning. A program must be reset before it can be run again.

+ **START**. The **START** command begins program execution in either the **BLOCK** mode or the **AUTO** mode. The **START** key is also used to restart the program after it has stopped for drilling, Z axis adjustment, or a programmed stop.

<cursor UP>
<cursor DOWN> FDR^{OVR} The FD^{OVR} commands are used to override the programmed feedrate. The <up arrow> key raises the feedrate by 5% and the <down arrow> key lowers the feedrate by 5% each time they are pressed. The feed override amount is shown as a percent of the programmed rate on the right side of the screen.

<cursor LEFT> LOAD The **LOAD** command lets the operator select a part program to be loaded. When a program is selected it is loaded into memory and becomes the active program.

<cursor RIGHT> VIEW This enables previewing the part-program by simulating the cutter path on the display screen. For more detailed information on VIEW see Chapter 12 in this manual.

* **EDIT** This key is used to edit the currently loaded part program.

/ **OPTS**. This key is used to change the setting of the **RUN OPTS** parameter and the **BLOCK DEL** parameter. These parameters enable/disable the OPTIONAL STOP and BLOCK DELETE instructions. See the chapter titled **MDI** in this manual for more information on the OPTIONAL STOP and BLOCK DELETE commands.

CHAPTER 10
UTILITIES

Introduction

This chapter discusses the use of the **UTILS** functions that are available on the EZ-PATH control.

Calling up the UTILS screen

The **UTILS** screen is called up when the **9 UTILS** key is pressed while in the BASIC OPERATIONS screen. When the FILE UTILS screen is called, the main EZ-PATH screen disappears and the commands menu shown below is displayed.

```
EZPATH UTILITIES Version: 2.6

     <1> COMPLETE DISKCOPY.

     <2> COPY files from EZPATH to FLOPPY DISK.
     <3> COPY files from FLOPPY DISK to EZPATH.

     <4> DELETE files

     <5> VIEW contents of file on FLOPPY DISK.
     <6> VIEW contents of file on EZPATH.

     <7> SEND or RECEIVE files.

     <ESC> QUIT to EZPATH.

>>> Select:
```

These commands are used to copy files back and forth between the system disk in memory, and the floppy diskette in the disk drive in the back of the machine, and other sources. Each of these commands is discussed below.

<1> COMPLETE DISKCOPY

Press **1** to copy a diskette. This requires swapping the diskettes at least three times. If you are copying the EZ-PATH system diskette, make sure the diskette is locked before copying. To lock a diskette, slide the small tab on the back of the diskette so that the hole is uncovered. This prevents the disk from being written on. **Remember**, the diskette must be unprotected to start the EZ-PATH (slide the tab so the hole is covered up).

```
<2> COPY files from EZPATH to FLOPPY DISK
```

Press **2** to copy **PGM** files from the hard disk drive to a floppy diskette for permanent storage.

NOTE: A 3.5 inch diskette must be in the disk drive before selecting any of these commands.

When **2** is selected, the following choices are shown on the screen.

```
COPY FILES. < Insert DISK in FLOPPY DRIVE>.

     <1> SHOW ALL EZPATH files on EZPATH.
     <2> SHOW ALL EZPATH files on FLOPPY DISK.

     <3> SHOW OTHER FILES on EZPATH.
     <4> SHOW OTHER FILES on FLOPPY DISK.

     <5> COPY ALL EZPATH files from EZPATH to FLOPPY DISK.
     <6> COPY ALL OTHER files from EZPATH to FLOPPY DISK.

     <7> COPY AN EZPATH file from EZPATH to FLOPPY DISK.
     <8> COPY OTHER file from EZPATH to FLOPPY DISK.

     <9> COPY TOOL LIBRARY from EZPATH to FLOPPY DISK.

     <ESC> EXIT

>>> Select:
```

Pressing the **1** key lists the PGM files on the EZPATH.
Pressing the **2** key lists the PGM files on the floppy diskette.
Pressing the **3** key lists the TXT files on the EZPATH.
Pressing the **4** key lists the TXT files on the floppy diskette.
Pressing the **5** key selects all PGM files in memory and copies them onto the floppy diskette.
Pressing the **6** key selects all TXT files in memory and copies them onto the floppy diskette.
Pressing the **7** key lets you choose one PGM file, by name, and copy it to the floppy disk.
Pressing the **8** key lets you choose one TXT file, by name, and copy it to the floppy disk.
Pressing the **9** key lets you choose the Tool Library, by name, and copy it to the floppy disk.

Pressing the **ESC** key exits this menu, and returns to the main **UTILS** screen.

```
<3> COPY files from FLOPPY DISK to EZPATH.
```

Press **3** to copy **PGM** files (EZ-PATH part program files) from the floppy diskette to the EZPATH memory.

To copy programs from a 3.5 inch diskette into memory, put the desired diskette in the drive, then select **3** from the main UTILS screen.

When **3** is selected, the following choices are shown on the screen.

```
COPY FILES. < Insert DISK in FLOPPY DRIVE>.

     <1> SHOW ALL EZPATH files on EZPATH.
     <2> SHOW ALL EZPATH files on FLOPPY DISK.

     <3> SHOW OTHER FILES on EZPATH.
     <4> SHOW OTHER FILES on FLOPPY DISK.

     <5> COPY ALL EZPATH files from FLOPPY DISK to EZPATH.
     <6> COPY ALL OTHER files from FLOPPY DISK to EZPATH.

     <7> COPY AN EZPATH file from FLOPPY DISK to EZPATH.
     <8> COPY OTHER file from FLOPPY DISK to EZPATH.

     <9> COPY TOOL LIBRARY from FLOPPY DISK to EZPATH.

     <ESC> EXIT

>>> Select:
```

Pressing the **1** key lists the PGM files on the EZPATH.
Pressing the **2** key lists the PGM files on the floppy diskette.
Pressing the **3** key lists the TXT files on the EZPATH.
Pressing the **4** key lists the TXT files on the floppy diskette.
Pressing the **5** key selects all PGM files on the diskette and copies them to the hard disk.
Pressing the **6** key selects all TXT files on the diskette and copies them to the hard disk.
Pressing the **7** key lets you choose one PGM file, by name, and copy it to the hard disk.
Pressing the **8** key lets you choose one TXT file, by name, and copy it to the hard disk.
Pressing the **9** key lets you choose the Tool Library, by name, and copy it to the hard disk.

Pressing the **ESC** key exits this menu, and returns to the main **UTILS** screen.

<4> DELETE FILES:

This command is used to delete **PGM** files from the diskette in the floppy drive.

When **4** is selected, the following choices are shown on the screen.

```
DELETE FILES. < Insert DISK in FLOPPY DRIVE>.

     <1> SHOW ALL EZPATH files on EZPATH.
     <2> SHOW ALL EZPATH files on FLOPPY DISK.

     <3> SHOW OTHER FILES on EZPATH.
     <4> SHOW OTHER FILES on FLOPPY DISK.

     <5> DELETE AN EZPATH file on EZPATH.
     <6> DELETE OTHER program file on EZPATH.

     <7> DELETE AN EZPATH file on FLOPPY DISK.
     <8> DELETE OTHER program file on FLOPPY DISK.

     <ESC> EXIT

>>> Select:
```

Pressing the **ESC** key exits this menu, and returns to the main UTILS menu.

Pressing the **1** key lists the PGM files on the EZPATH.
Pressing the **2** key lists the PGM files on the floppy diskette.
Pressing the **3** key lists the TXT files on the EZPATH.
Pressing the **4** key lists the TXT files on the floppy diskette.
Pressing the **5** key deletes all PGM files on the hard disk and from the floppy diskette.
Pressing the **6** key deletes all TXT files on the hard disk and from the floppy diskette.
Pressing the **7** key lets you choose one PGM file, by name, and delete it from the hard disk, and from the floppy disk..
Pressing the **8** key lets you choose one TXT file, by name, and delete it from the hard disk, and from the floppy disk.

Pressing the **ESC** key exits this menu, and returns to the main **UTILS** screen.

<5> VIEW contents of file ON FLOPPY DISK.

Press **5** to display the program contained in a file on the floppy disk.

When **5** is selected, the following choices are shown on the screen.

```
VIEW / SHOW files. <Insert FLOPPY DISK in DRIVE>

     <1> Show ALL EZPATH files on FLOPPY DISK.
     <2> View AN EZPATH file on FLOPPY DISK.

     <3> Show ALL OTHER files on FLOPPY DISK.
     <4> View OTHER files on FLOPPY DISK.

     <ESC> EXIT

>>> Select:
```

Pressing the **1** key lists the **.PGM** files that are on the floppy disk in the disk drive.
Pressing the **2** key lets you select a **.PGM** file, and then displays the program in this file on the EZ-PATH screen.
Pressing the **3** key lists the **.TXT** files on the floppy disk in the disk drive.
Pressing the **4** key lets you choose one **.TXT** file, and then displays the program in this file on the EZ-PATH screen.

Pressing the **ESC** key exits this menu, and returns to the main Utils screen.

<6> VIEW contents of file ON EZPATH.

Press **6** to display the program contained in a file on the EZPATH hard disk.

When **6** is selected, the following choices are shown on the screen.

```
VIEW / SHOW files. <Insert FLOPPY DISK in DRIVE>

     <1> Show ALL EZPATH files on EZPATH.
     <2> View AN EZPATH file on EZPATH.

     <3> Show ALL OTHER files on EZPATH.
     <4> View OTHER files on EZPATH.

     <ESC> EXIT

>>> Select:
```

Pressing the **1** key lists the **.PGM** files that are in the EZ-PATH's memory.
Pressing the **2** key lets you select a **.PGM** file, and then displays the program in this file on the EZ-PATH screen.
Pressing the **3** key lists the **.TXT** files on the EZ-PATH's hard disk.
Pressing the **4** key lets you choose one **.TXT** file, and then displays the program in this file on the EZ-PATH screen.

Pressing the **ESC** key exits this menu, and returns to the main Utils screen.

```
<7> SEND or RECEIVE files.
```

This utility is used to communicate with a remote device, such as a computer running EZ-CAM. The remote device is connected to the EZ-PATH by a cable that is plugged into an RS232 port (COMM 1 or COMM2) on the remote device and into the serial port of the machine. Bridgeport Machines EZ-CAM adapter cable (part no. 1940515) and Universal Communications cable (part no. 1940303) can be used for this purpose.

The cable configuration necessary to communicate to the EZ-PATH is a **null modem cable** configured as follows:

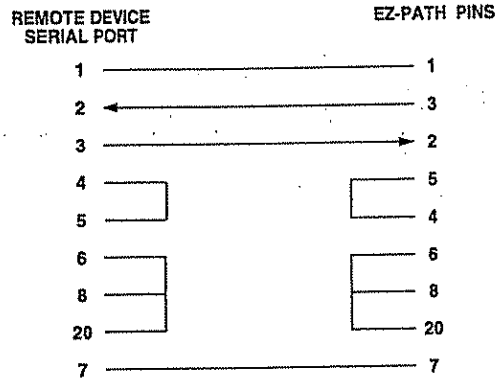


Figure 10-1

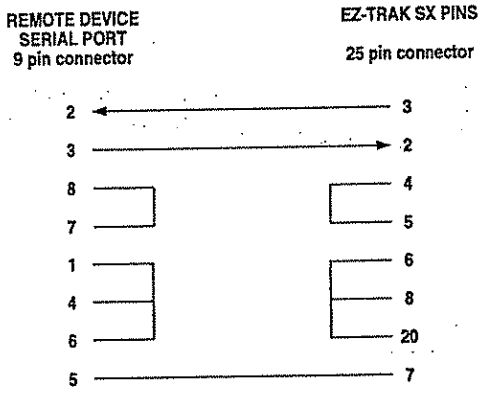


Figure 10-2

The EZ-PATH supports three different communications protocols. These are **EZ-LINK**, **YMODEM**, and **ASCII**.

When 7 is selected from the main UTILS screen the following commands are displayed.

```
SEND-RECEIVE Protocol

     <1> EZ-LINK.
     <2> YMODEM.
     <3> ASCII.

     <ESC> QUIT.

>>> Select:
```

Pressing the **1** key selects **EZ-LINK**, the communications format used by Bridgeport equipment.

When the **EZ-LINK** format is chosen, the screen displays:

```
EZ-Link CONNECTED To:

     <1> EZ-PATH.
     <2> EZ-CAM.
     <3> EZ-FILE.

     <ESC> QUIT.

>>> Select:
```

Select the type of device that your EZ-PATH machine is connected to. Do **not** select **EZPATH** unless the program is being sent from one EZ-PATH machine to another.

Selecting the **EZPATH** as the other device displays the following menu:

```
EZ-Link MASTER \ SLAVE:

     <1> MASTER.
     <2> SLAVE.

     <ESC> QUIT.

>>> Select:
```

The EZ-Link format is designed so that one device (the CNC control) can send files to, or request files from, the other device (usually a computer). This is called the **MASTER** device.

The computer running EZ-CAM Utils is a passive device, it cannot select a file to send or receive, it only reacts to the commands of the CNC. This is called the **SLAVE** device.

When the EZ-PATH machine is connected to a computer (running EZ-CAM Utils) or an EZ-FILE device, the EZ-PATH is automatically the MASTER, and the computer or EZ-FILE is the SLAVE. (If you are using an EZ-FILE, see **USING AN EZ-FILE**, in this chapter.)

When one EZ-PATH is connected to another EZ-PATH, one must be chosen as the MASTER and the other as the SLAVE. The commands to send and receive are chosen only at the MASTER EZ-PATH machine. Be sure to make one machine the MASTER and the other machine the SLAVE.

SENDING AND RECEIVING FILES

Regardless of the Communications Protocol selected, the same commands are used to send and receive files. The following list of commands is applicable to all three communications protocols.

```
SEND-RECEIVE Programs.

     <1> Send EZPATH file.
     <2> Send OTHER file.

     <3> Receive EZPATH file.
     <4> Receive OTHER file.

     <5> Show ALL EZPATH files on EZPATH.
     <6> Show ALL OTHER files on EZPATH.

     <ESC> EXIT.
```

Pressing the **1** key readies the EZ-PATH to send a **.PGM** file to the remote computer. You are prompted to enter the name of the program to send, and then the EZ-PATH waits for the remote computer to signal it is ready to receive.

Pressing the **2** key readies the EZ-PATH to send a **.TXT** file to the remote computer. You are prompted to enter the name of the program to send, and then the EZ-PATH waits for the remote computer to signal it is ready to receive.

Pressing the **3** key readies the EZ-PATH to receive a **.PGM** file. You are prompted for the name of the file that will be sent, then the EZ-PATH waits for the remote computer to begin sending.

Pressing the **4** key readies the EZ-PATH to receive a **.TXT** file. You are prompted for the name of the file that will be sent, then the EZ-PATH waits for the remote computer to begin sending.

Pressing the **5** key lists the **.PGM** files that are already stored in the EZ-PATH's memory.

Pressing the **6** key lists the **.TXT** files that are already stored in the EZ-PATH's memory.

COMMUNICATING WITH THE EZ-PATH

All three of the communications protocols available on the EZ-PATH can be used to both send and receive a file from a remote computer system. The remote system should be running either a communications program that supports one of the three protocols (EZ-LINK, YMODEM, or ASCII) or the EZUTILS.EXE program from the EZ-PATH System Diskette. This program runs under DOS, and supports communications on any PC through a standard **COMM1 ONLY** serial communications port. Any of the three protocols may be used if the remote computer is running this program.

NOTE: When using the EZ-LINK protocol, the SLAVE device (EZ-CAM Utils, EZ-FILE, or a computer running EZUTILS.EXE) should be set on-line first. The EZ-PATH machine (the MASTER device) must be set on-line second.

Receiving Files on the EZ-PATH

This procedure is the same, regardless of what communications protocol is used, and regardless of the remote computer's type and software.

At the EZ-PATH:

1. Press the **UTILS** key from the basic operations screen.
2. Select `SEND or RECEIVE files.`
3. Select the communications protocol that is supported by the remote computer.

At the Remote Computer:

1. Run the communications program.
2. Prepare the computer to send a file using the correct protocol. Be sure that the file is named with a number (up to seven digits are allowed) and the extension, either **.PGM** (if the file is in the EZ-PATH's language) or **.TXT** (for a G-code file).

At the EZ-PATH:

1. Select either `Receive EZPATH file` or `Receive OTHER file` depending on the file type.
2. Enter the name of the file as it should be saved on the EZ-PATH. It is not necessary or possible to enter the file extension. Only the file name is required.

At the Remote Computer: (This step is not necessary when using EZ-LINK.)

1. Select the command to send the file.

USING AN EZ-FILE

Sending a program to an EZ-FILE

This procedure is used when a file is being sent to an EZ-FILE device from an EZ-PATH.

At the EZ-PATH:

1. Press the **UTILS** key from the basic operations screen.
2. Select SEND or RECEIVE files.
3. Select the EZ-LINK communications protocol.
4. Press the **3** key to choose the EZ-FILE device.
5. Select **1** or **2** depending on the file you want to send.
6. Enter the name of the file to send. Press the **ENTER** key.

At the EZ-FILE:

1. Turn the EZ-FILE **ON**.
2. Enter the Channel Number (47 for channel 0, or 48 for channel 1) followed by #.
3. Press **4 #** to select the EZ-LINK protocol.
4. Press **1 #** to define a file.
5. Enter the file name and press the # key. The disk light goes on for a moment and then the FILE OPEN light blinks. The EZ-FILE is ready to receive.

The ACTIVITY light on the EZ-FILE should light while the data is being sent. When the transfer is complete only the POWER light remains on.

Receiving a program from an EZ-FILE

This procedure is used when a file is being sent to an EZ-PATH machine form an EZ-FILE.

At the EZ-PATH:

1. Press the **UTILS** key from the basic operations screen.
2. Select `SEND or RECEIVE files.`
3. Select the EZ-LINK communications protocol.
4. Press the **3** key to choose the EZ-FILE device.
5. Select **3** or **4** depending on the file you want to receive. (All files on the EZ-FILE are stored as TXT files. You will have to keep track of file names and types.)
6. Enter the name of the file to receive.

At the EZ-FILE:

1. Turn the EZ-FILE **ON**.
2. Enter the Channel Number (47 for channel 0, or 48 for channel 1) followed by #.
3. Press **4 #** to select the EZ-LINK protocol.
4. Press **2 #** to select a file.
5. Enter the file name of the file and press the # key. The disk light goes on for a moment and then the FILE OPEN light blinks. The EZ-FILE is ready to send.

CHAPTER 11
TEACH MODE

INTRODUCTION

The **TEACH** mode of the EZ-PATH is a simple way of saving a part program as it is performed manually. The operator cuts the part manually using the EZ-PATH in the TEACH mode and saves a point at the end of each move. The EZ-PATH stores each point and can save the entire program for later recall and editing. The points (0–99) are saved as they are created in a file called TEACH.PGM.

NOTE: When created, the points file is stored as a file called TEMP.TXT and as an editable file called TEACH.PGM. If desired, this file can be renamed and saved on a 3.5" diskette for future re-use, as a program file.

To enter the **TEACH** mode press the **. SAVE PT** key in the BASIC OPERATIONS screen, or press **. SAVE PT** in the **JOG** screen. The screen displays:

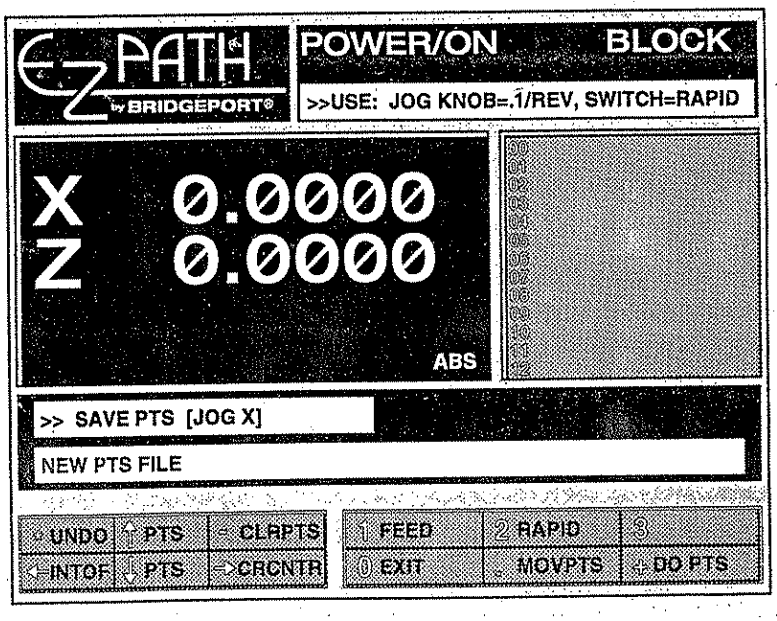


Figure 11-1

The commands available from the TEACH screen are:

1 FEED Press the **1** key to save the current point as a **FEED EVENT**. This means that the EZ-PATH will move in a straight line to this point from the previous location.

2 RAPID Press the **2** key to save the current position as a **POSITION EVENT**. The EZ-PATH will make a positioning move to this point when the points are replayed, at the rapid speed.

3 JOG X or **JOG Z**. The **3** key in the TEACH mode may read **JOG X** or **JOG Z** or it may appear blank. Its appearance depends on how the TEACH mode was reached. If the TEACH mode was entered from the **JOG Z** or **JOG X** screen, the **3** key will select the other axis to **JOG**. If TEACH was selected from the BASIC OPERATIONS screen, then the **3** key has no function, and it appears blank.

+ DO POINTS This key replays the saved points in the same order that they were saved.

. MOVPTS The **.** key is used to move the tool position to a specified point. When this key is selected, a dialog box appears in the upper left corner of the screen. It prompts the operator to enter the number of the point that the machine will move to. After the point is selected, press the ENTER key.

– CLR POINTS The – key deletes all of the saved points from the points file.

*** UNDO** The * key deletes only the last saved point from the points file. The previous point is then displayed on the screen. The **UNDO** command can be used repeatedly to delete several points.

ESC The **ESC** key exits the **TEACH** mode. When the **TEACH** mode is exited, the programmed points are saved in a file called **TEACH.PGM**.

<left arrow> INTOF This command creates a point at the intersection of two lines that are defined each with two points that are already in the points list. These four points must be consecutive in the list. When this command is selected, the user is prompted for the number of the first of the four points in the list. The user is then shown the coordinates of the new point and the system prompts for the number where this point is to be stored.

<right arrow> CRCNTR This command finds the center of a circle that is defined with three consecutive points in the points list. This point can then be stored in the points list as a separate point. When this command is selected, the user is prompted for the number of the first of the three points in the points list. The user is then shown the coordinates of the new point, and the system prompts for the number where this point is to be stored.

<up arrow> PTS This command scrolls the points list upward. The top of the points list is the 00 point.

<down arrow> PTS This command scrolls the points list downward. The bottom of the points list is the point 99.

CHAPTER 12
PREVIEW MODE

INTRODUCTION

The **PREVIEW** mode is used to execute and pre-view the cutter path of a part program on the CRT display before actually running the program. The screen shows an XZ view of the part and displays tool movements on the screen so that the program can be checked before executing the program.

The **PREVIEW** mode is selected from the **RUN** screen. NOTE: A part program should be loaded into memory (via the **LOAD** command) **before** selecting the **VIEW** command from the **RUN** screen.

NOTE: The **VIEW** command in the **RUN** screen displays actual tool movements, as they are generated by the BMDC (Bridgeport Motor Drive Controller). This is different from the **VIEW** command in the **EDIT** or **MDI** mode, which shows only part geometry.

THE PREVIEW SCREEN

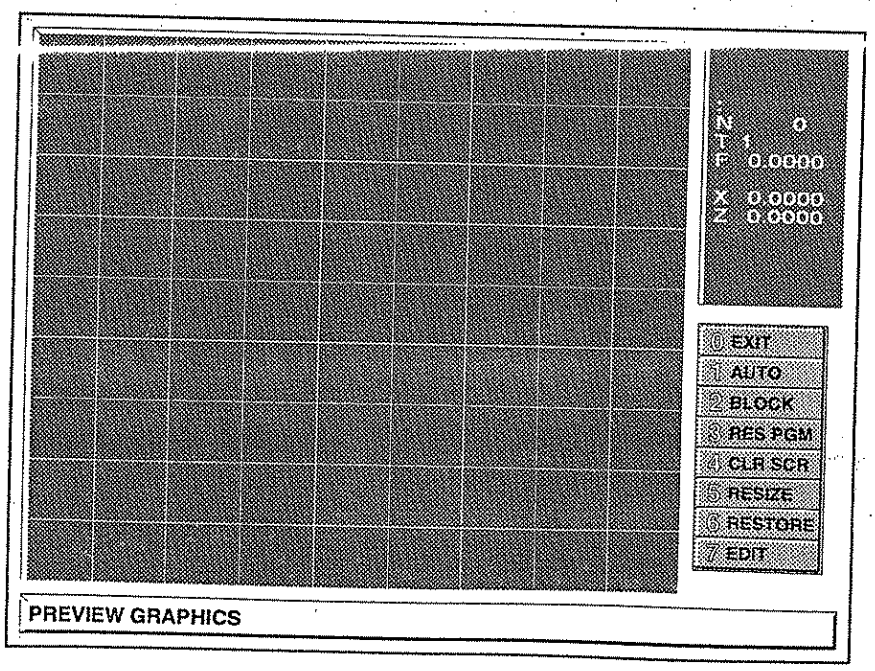


Figure 12-1

KEY FUNCTIONS

0 EXIT The **EXIT** command leaves the **PREVIEW** mode and returns to the RUN screen.

1 AUTO The **AUTO** command sets the loaded program to be run in continuous operation. The preview halts for programmed stops, or tool changes. The program begins when the **AUTO** key is pressed.

2 BLOCK The **BLOCK** commands sets the loaded program to be run in single step mode. Each line of a program is executed, and the program is halted until the operator presses the **2 BLOCK** key. The program run begins when the **2 BLOCK** key is pressed.

3 RES PGM The **RES PGM** command resets the program back to the beginning. It can then be re-started from the beginning. It is usually a good idea to use the **CLR SCR** command before re-running a program.

4 CLR SCR The **CLR SCR** command clears the screen. The **PREVIEW** area is erased so that the program can be seen more clearly. Once the screen is cleared, the erased information cannot be redrawn, except by running the program again.

5 RESIZE The **RESIZE** command is used to change the size of the PREVIEW window. When the **RESIZE** command is selected from the PREVIEW screen, the operator is prompted to enter four numbers which control how the PREVIEW screen is displayed. Prompts are displayed for:

Xmin	the far edge of the part
Zmin	the left edge of the part
Xmax	the near edge of the part
Zmax	the right edge of the part

The **Xmin** and **Zmin** values can be set to the far left corner of the part. The **Xmax** and **Zmax** values can be set to the near right corner of the part.

These values determine the window size of the PREVIEW screen. These numbers are altered slightly to fill the screen area. It is generally a good idea to increase these values so that the part is displayed more towards the center of the screen.

Other prompts for **RESIZE** are:

SHOW TOOL	will show the tool shape when turned on
SHOW PATH	will show the tool path when turned on
TRACKING POINT	will track the tool either from the touch off point or the tool center

6 RESTORE The **RESTORE** command returns the PREVIEW screen to the original size set by the EZ-PATH, after the screen size has been changed using the **RESIZE** command.

7 **EDIT** The **7 EDIT** key in the PREVIEW mode calls the EDIT mode, and allows the operator to make changes to the program. The program must be saved, if changes are made. If EDIT is invoked from the PREVIEW mode, it will not automatically return to the PREVIEW mode.

USING PREVIEW

When the **VIEW** command is selected, the program is automatically scanned, and the **VIEW** window is sized accordingly. The screen displays:

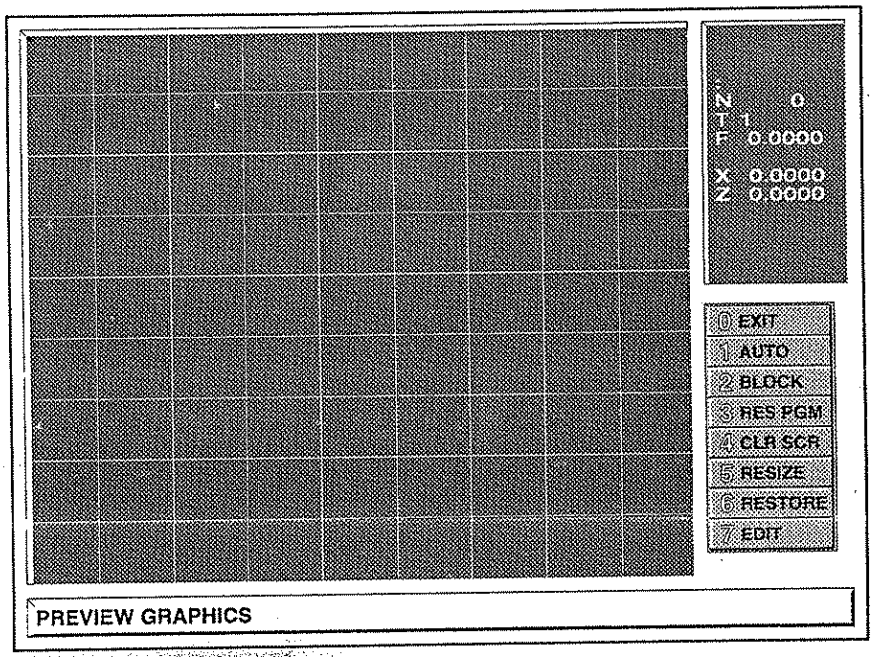


Figure 12-2

Press the **1 AUTO** key to begin simulation of the part program on the screen. The **START** button must also be pressed for tool changes, or programmed stop events, just as if the program were actually executing on the machine.

Cutting moves are shown on the screen as solid lines. Rapid moves are shown as dotted lines. The part shape, or PATH (the programmed PATH) is shown on the screen before the tool movements are executed. The tool shape is also automatically shown on the screen with the tool movements. The PATH and tool shape can be turned on and off with the **5 RESIZE** screen. Enter a zero (0) in the appropriate box to shut off the desired function. Also, the TRACKING PT of the tool can be changed from either the Touch Off Pt (0) or the Tool Center (1).

To zoom in on a particular location, use the **RESIZE** command to change the window settings. Use the **RESTORE** command to return the window to its original settings.

Appendix A
EZ-PATH System Overview

Introduction

This section includes:

- A description of the EZ-PATH computer hardware
- A description of the EZ-PATH System Software
- A description of how the EZ-PATH handles USER DATA

The EZ-PATH Control Hardware

The EZ-PATH control contains two micro-processor based systems. The first is an 80386 based PC AT micro-computer which runs the operator interface displayed on the computer screen. The second micro-processor based system is a 32-bit 68030 based Bridgeport-designed Motor Drive Controller (BMDC) which runs the real time system tasks.

Unlike competitive PC based CNC systems, which are loosely coupled via serial or parallel links, the EZ-PATH is a true PCNC system– the BMDC board is plugged directly into the PC data bus so the two systems communicate via shared memory. When combined with the on-board 96 bit floating-point processor, the control leaves older 16 bit based CNC systems way behind in terms of computing power.

Standard throughput in the EZ-PATH is 250 data blocks per second. 256 kilo-bytes of high speed memory (equal to more than 2000 ft. of paper tape) is provided on the BMDC board for part program storage. In addition to the 4 mega-byte of RAM used by DOS and Bridgeport Operating System on the PC AT, an additional 170 mega-byte hard drive is provided for data storage.

A 9" monochrome VGA monitor, pop-up windows, and soft keys make control operation quick and easy. All machine functions are selected via the keypad which is linked to on-screen operator commands.

EZ-PATH SOFTWARE

The software which drives the EZ-PATH is best described in terms of the hardware that runs it. All of this software is loaded from the hard disk at startup.

PC AT. The software used on the PC AT consists primarily of the Microsoft DOS v. 6.2 operating system, and EZPATH.EXE, the operator interface software.

> **DOS** (disk operating system). DOS is a collection of routines that perform basic computer tasks such as starting the computer (booting up), moving data to and from disks and peripheral devices, and managing and allocating memory space.

On the EZ-PATH control, DOS is the bottom layer of the software that runs on the PC AT. In normal operation, DOS is not visible to the user. On start-up, a routine called AUTOEXEC.BAT automatically loads the EZ-PATH software into the system and starts it up. For more information on DOS, refer to the MS-DOS User's Reference Manual.

EZPATH.EXE. This software contains the routines that the operator uses to run the EZ-PATH. These routines call up the display screens, execute the commands the operator selects and communicate with the BMDC board to update system status.

BMDC Motor Drive Controller.

BMDC.BIN. These routines provide real time control of the system and include the part program parser and executor, 2 axes of servo drive control, interpolation algorithms, and system monitor.

SYSTEM DISK. The SYSTEM DISK is a 3.5" high-density floppy disk that stores all of the software for the EZ-PATH. This software is duplicated on the EZ-PATH hard disk; however, the SYSTEM DISK is the **only** backup available in case of catastrophic loss of the data on the hard disk.

The SYSTEM DISK is not needed on a day-to-day basis under normal operations. The system software and parameter files are loaded from the hard disk drive each time the machine is powered on. User part programs are also stored there.

It is highly recommended that you keep the SYSTEM DISK in a safe place.

NOTE: Certain parameter files unique to each machine are stored on the SYSTEM DISK. This includes axes backlash values and machine parameter files.

EZ-PATH SYSTEM SOFTWARE. The following files make up EZ-PATH system software:

```
EZPATH      EXE   EZPATH software
BMDC        BIN   BMDC software
BMDCPRMS    SYS   parameter file
RESET       EXE   program that allows resetting without turning off
EZLOAD      EXE   program that loads BMDC.BIN
METWNDO3    DRP   required protected mode driver
METWNDO5    DRP   required protected mode driver
TOOLLIB           Tool Library file
OFFSETS           Offsets file
TLICN1      ICN   Tool Library icon file
TLICN2      ICN   Tool Library icon file
TLICN3      ICN   Tool Library icon file
TLICN4      ICN   Tool Library icon file
EZTRAK      ICN   Geometry Help icons
PWRFD       ICN   Illustration for Power Feed
CHAMFER     ICN   Illustration for Chamfer
TAPER       ICN   Illustration for Taper
BLD008      FNT   Required font file
```

BLD013	FNT	Required font file
BLD019B	FNT	Required font file
BLD019M	FNT	Required font file
HLV025B	FNT	Required font file
HLV050B	FNT	Required font file
SYSTEM08	FNT	Required font file
SYSTEM16	FNT	Required font file
SYSTEM00	FNT	Required font file
SYSTEM72	FNT	Required font file
VG8_FONT	AHS	Required font file
VGA_FONT	AHS	Required font file
EZPATH	PCX	EZPATH logo file

USER DATA

The hard disk also contains USER created part programs. Two types of files can be used on the EZ-PATH. Files with the extension **.PGM** are created on EZ-PATH machines. Files with the extension **.TXT** are compatible with other Bridgeport controls. Both of the these types of programs may be found on the EZ-PATH hard disk.

USER DATA PATHS

There are three storage areas for data in the EZ-PATH. Understanding how the system handles data is necessary to properly operate the EZ-PATH.

1. A floppy disk containing part programs is accessed from the floppy disk drive mounted in the control cabinet. The designation of the floppy drive is **A:**.

2. The system contains a fixed hard disk so that the control has fast data access. System software is automatically loaded from the hard disk to the BMDC at START-UP. Programs are automatically saved to the hard disk when they are SAVED by the operator. The designation of the hard disk is **C:**.

   If it is necessary to load a part program from the floppy disk (**A:**) to the hard disk (**C:**), then the **UTILS** command is used.

3. Before a part program can be executed, it must be loaded into the BMDC memory. The **LOAD** command transfers a part program to this area of memory. The **SAVE | RUN** command in the MDI mode will also automatically load the edited program into the BMDC memory area.

Appendix B
GEOMETRY HELP

Introduction

This section of this manual discusses the use of the **GEOMETRY HELP** functions that are available in many of the DO EVENT and MDI commands in the EZ-PATH software. Each of the commands in the GEOMETRY HELP menu is discussed in this section, and is detailed with several illustrations.

Calling up the GEOMETRY HELP screen

The **GEOMETRY HELP** screen is called up when the **/GEO** button (the / key) is pressed while entering data into one of the DO EVENT or MDI commands. When the GEOMETRY HELP screen is called, the screen displays the menus shown in Figure B-1.

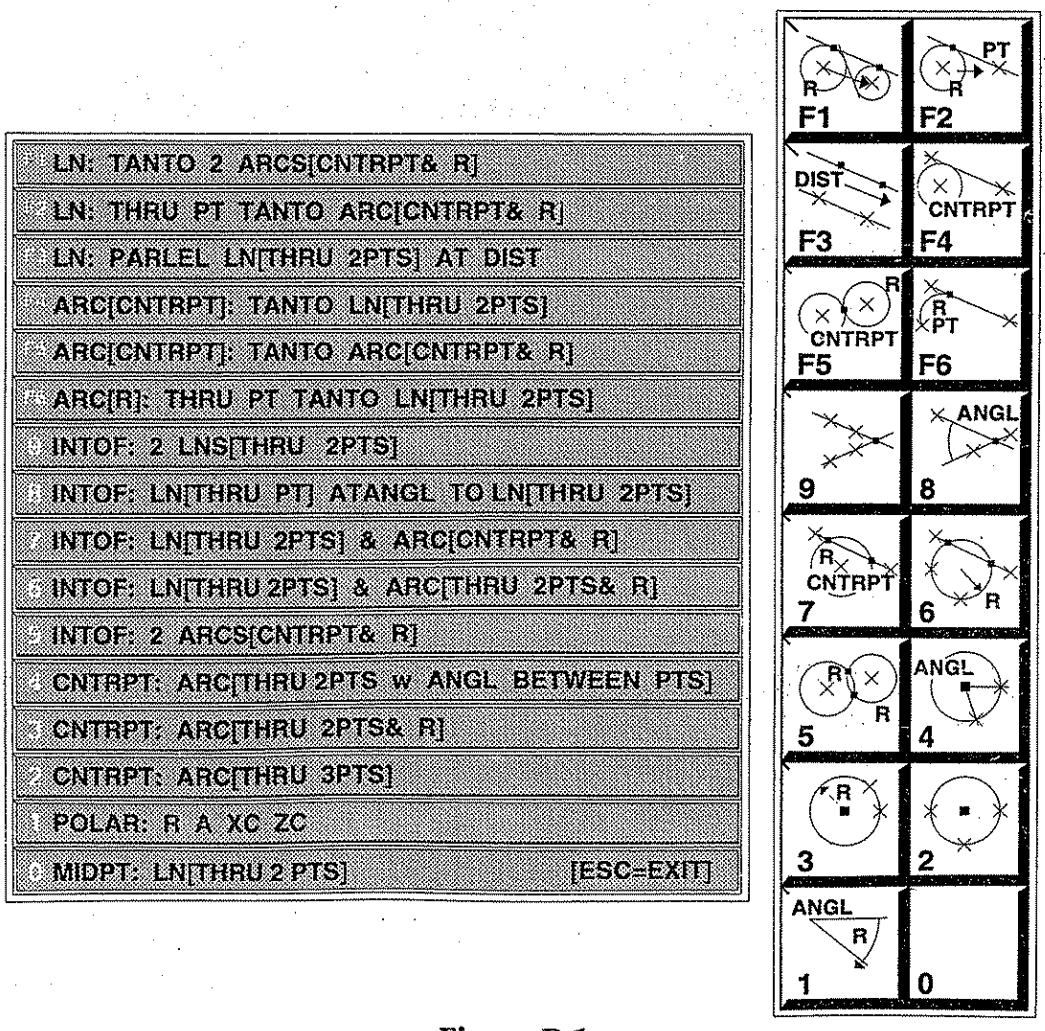


Figure B-1

The left side of this menu lists each of the GEOMETRY HELP commands, and the appropriate key which calls each one. The right side of the menu gives a graphic image of each command and the data it can calculate. The appropriate key is also listed with each of these graphic images.

USING GEOMETRY HELP

Drawings from which parts are machined are not always dimensioned with all the data necessary to make a part program. To help with this situation, the EZ-PATH has a GEOMETRY HELP function which automatically calculates the coordinates of the geometry most frequently used in part drawings. This mode is called by pressing the / key whenever coordinate data is requested.

The needed function is selected by pressing one of the keys that corresponds to a function on the screen. The function data box is then shown on the screen, and the required data must be entered.

GEOMETRY HELP COMMANDS

There are fifteen different GEOMETRY HELP commands. Each command is listed in this section with illustrations and examples, in the order they appear in the menu.

F1: LN: TANTO 2 ARCS [CNTRPT & R]

This command calculates the intersection points of a line which is tangent to two arcs, specified by their center points and radius values.

NOTE: There are four different lines that can be described as tangent to any two arcs. An example of this is shown in Figure B-2.

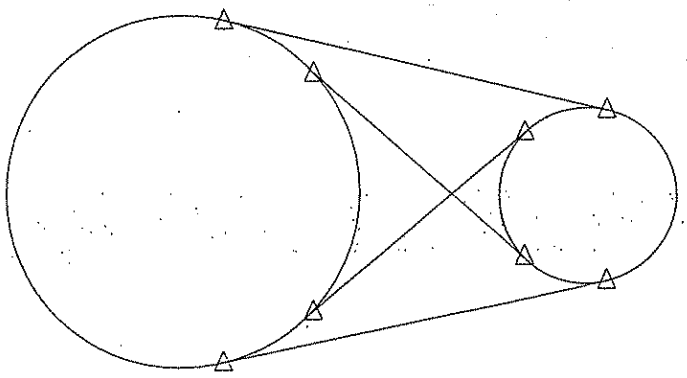


Figure B-2

The two lines which intersect between the two arcs are described as one set of lines (Figure B-3), and the two lines which do not intersect between the arcs are described as a different set of lines (Figure B-4).

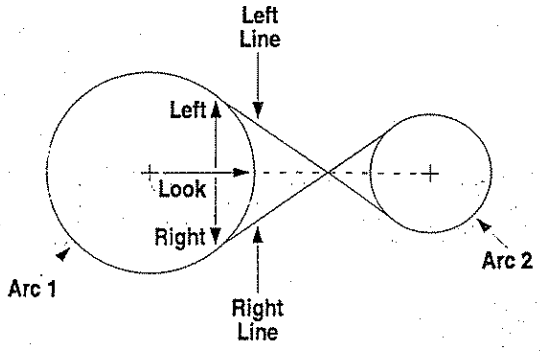


Figure B-3

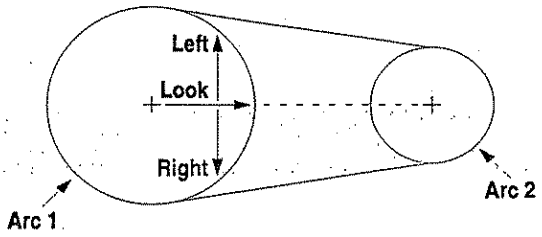


Figure B-4

Each set of two lines, (intersecting and non-intersecting) has a line on the left and a line on the right. These are determined by looking from the center of the first arc to the center of the second arc. See Figures B-3 and B-4.

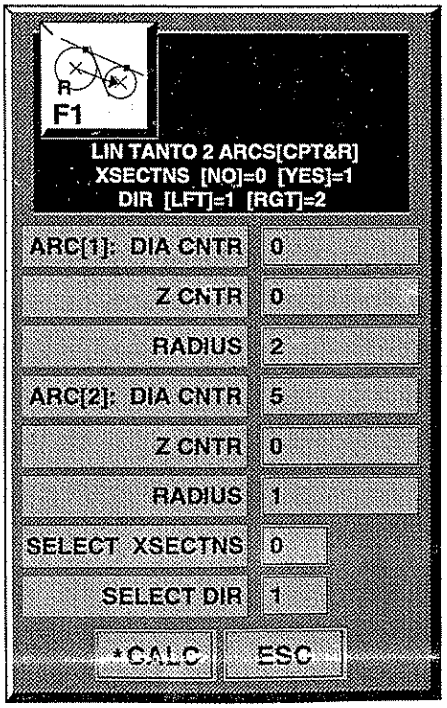


Figure B-5

ARC [1]:

DIA Cntr The X diameter of the center point of arc 1.

ZCntr The Z coordinate of the center point of arc 1.

Radius The radius value of arc 1.

ARC [2]:

DIA Cntr The X diameter of the center point of arc 2.

ZCntr The Z coordinate of the center point of arc 2.

Radius The radius value of arc 2.

Select XSECTNS This selects the set of intersecting lines or non-intersecting lines. See Figures B-3 and B-4.

Select DIR This selects the line on the left or right. [1=left 2=right]

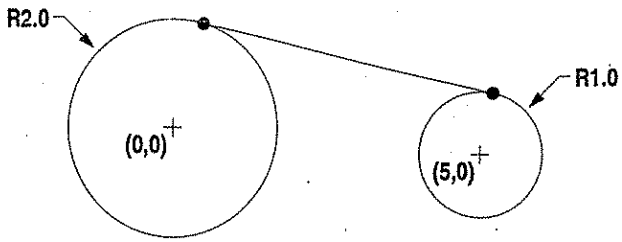


Figure B-6

When all of the data is entered, the system calculates the two intersection points for the chosen line. The coordinates of the two points are displayed as shown in Figure B-7. One of these points can be chosen as a location in a program, or a DO EVENT command. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure B-7. The chosen point coordinates are entered automatically into the correct data fields of the command previously selected, when the **Enter** key is pressed.

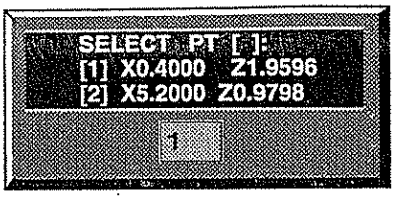


Figure B-7. The Calculated Intersection Points (See Figure B-6)

F2: LN: THRU PT TANTO ARC [CNTRPT & R]

This command calculates the intersection point of an arc and a line that is tangent to the arc. The line is defined by identifying the endpoint of the line which is not on the arc (shown by an X in Figure B-8). The arc is defined by its centerpoint, and radius. Figure B-8 shows how the line and arc are defined.

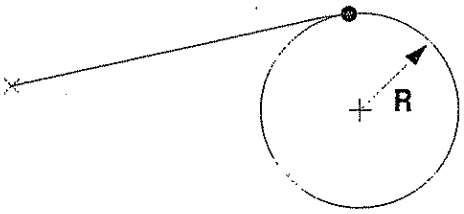


Figure B-8

NOTE: There are two different lines that can be defined through a point and tangent to an arc.

The two lines fall on either side of the arc. These lines are specified as being on the left and right of the arc. The direction is determined by looking from the center of the arc towards the specified point, as shown in Figure B-9.

The correct line is chosen by entering the direction in the **Select DIR** parameter. Enter a **1** for the direction if the line is on the **left**, or enter a **2** if the line is on the **right**.

In the example shown in the **F2** window (Figure B-10) the direction of the line is **right** (See Figure B-11). The **Select DIR** parameter has a value of **2** because the line is on the right.

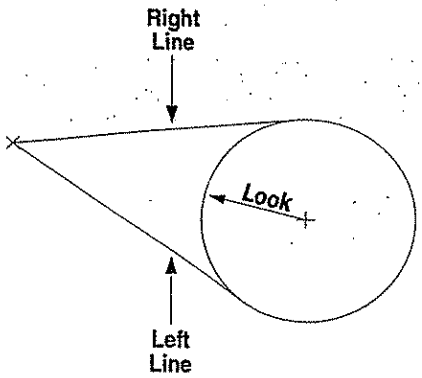


Figure B-9

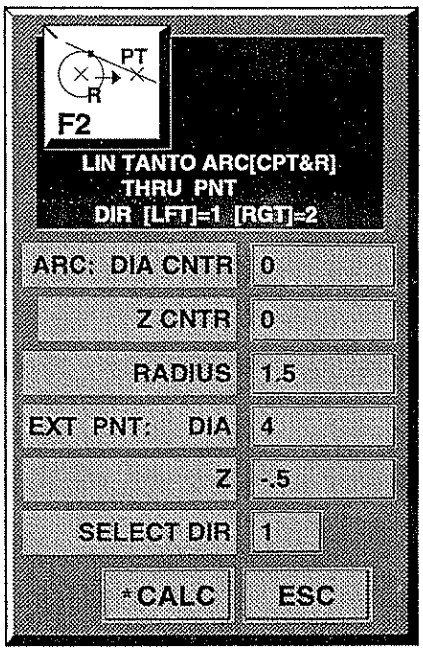


Figure B-10

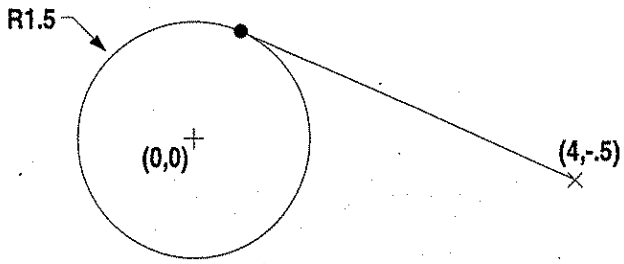


Figure B-11

ARC :	**DIA Cntr**	This is the X diameter of the center point of the arc.
	ZCntr	This is the Z coordinate of the center point of the arc.
	Radius	This is the radius value of arc 1.
EXT PNT:	**DIA**	This is the X diameter of the external point.
	Z	This is the Z coordinate of the external point.
Select DIR		This selects the line on the left or right. The direction is set by looking from the center of the arc towards the external point Enter **1** to select the line on the **left**, enter **2** to choose the line on the **right**.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection point for the chosen line. The point is entered automatically into the correct data fields of the command.

F3: LN PARLEL LN [THRU 2 PTS] AT DIST

This command calculates two points which are a given distance from a line. The line is defined by two points. The two calculated points form a line which is parallel to the given line. Figure B-12 shows the given points with an X, and the calculated points as •.

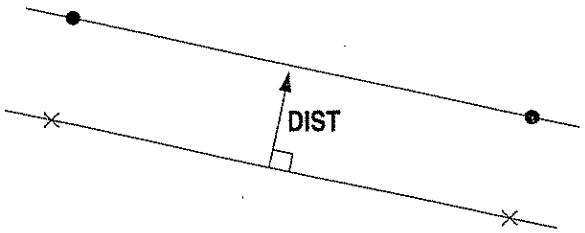


Figure B-12

NOTE: There are two lines that can be a specific distance from a given line, and be parallel to it.

The new line must be selected by giving a direction either left or right, from the given line. The direction is determined by looking from the first point of the given line to the second point of the given line. This is shown in Figure B-13 by the arrow at point 1.

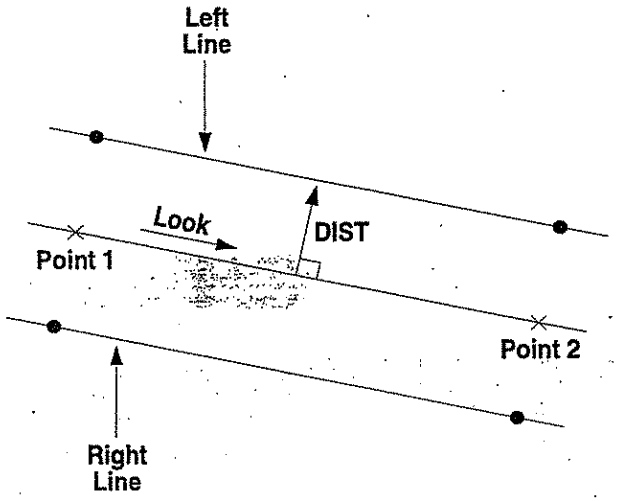


Figure B-13

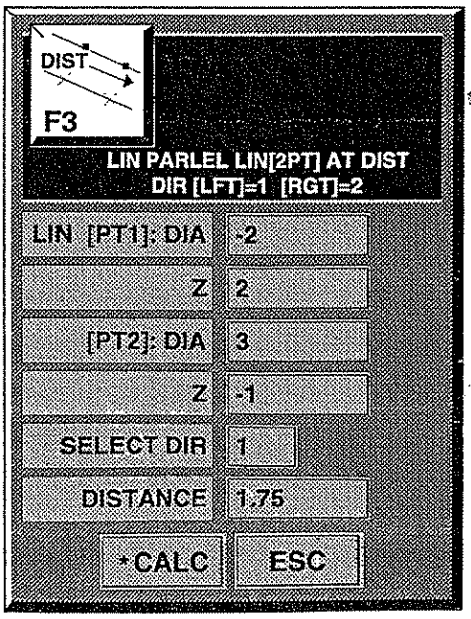


Figure B-14

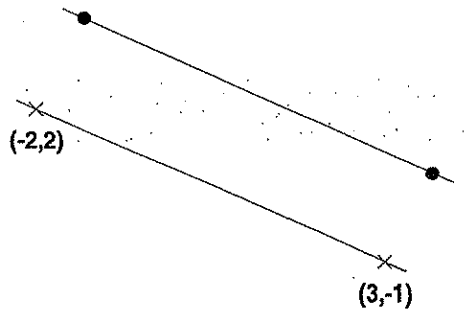


Figure B-15

NOTE: In the example in Figure B-15, the direction is **left**. The first point entered (shown in Figure B-14) is (-2,2).

LIN	**[PT1]:**	**DIA**	This is the X diameter of the first point of the line.
		Z	This is the Z coordinate of the first point of the line
	[PT2]:	**DIA**	This is the X diameter of the second point of the line.
		Z	This is the Z coordinate of the second point of the line.

Select DIR — This selects the line on the left or right. The direction is set by looking from the first point of the line towards the second point Enter **1** to select the line on the **left**, enter **2** to choose the line on the **right**.

Distance — This sets the distance away from the given line that the new line is placed.

When all of the data is entered, the system calculates the two points for the new line. The coordinates of the two points are displayed as shown in Figure B-16. One of these points can be chosen as a location in a program, or a DO EVENT command. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure B-17. The chosen point coordinates are entered automatically into the correct data fields of the command previously selected, when the **Enter** key is pressed.

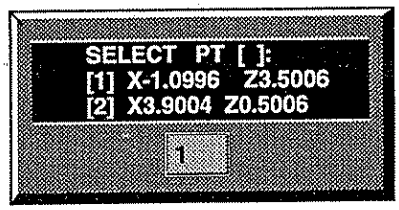


Figure B-16

F4: ARC [CNTRPT] : TANTO LN [THRU 2 PTS]

This command finds the intersection point of an arc and a tangent line. The centerpoint of the arc, and two points on the line must entered. Figures B-17 and B-18 show examples of how the line and the arc might be placed.

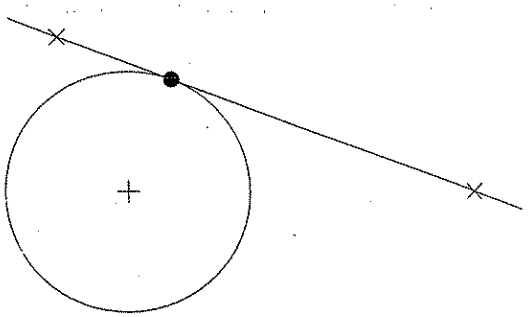


Figure B-17

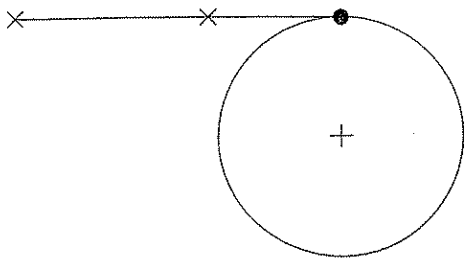


Figure B-18

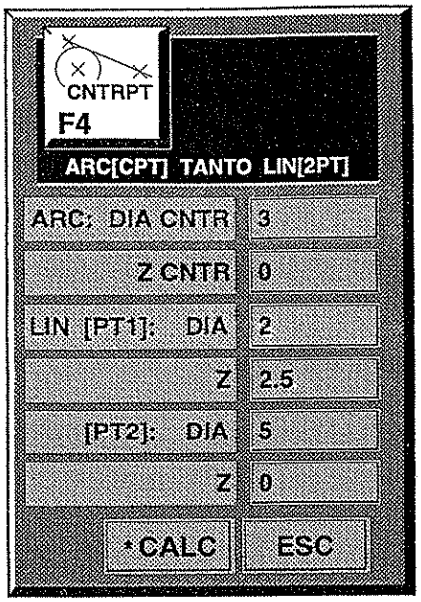


Figure B-19

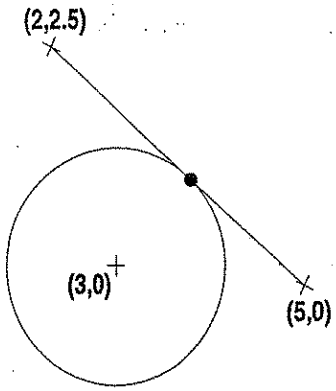


Figure B-20

ARC: **DIA Cntr** This is the X diameter of the arc's centerpoint.
ZCntr This is the Z coordinate of the arc's centerpoint.

LIN [PT1]: **DIA** This is the X diameter of the first point of the line.
Z This is the Z coordinate of the first point of the line.

LIN [PT2]: **DIA** This is the X diameter of the second point of the line.
Z This is the Z coordinate of the second point of the line.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection of the line and the arc, and the radius value of the arc. The point is entered automatically into the correct data fields.

F5: ARC [CNTRPT] : TANTO ARC [CNTRPT & R]

This command finds the intersection point of an arc, defined by its centerpoint, and a tangent arc, defined by its centerpoint and radius. Figure B-21 shows how the two arcs are defined. Remember, that the calculated point in this command is the intersection of the two arcs, shown in Figure B-21 as a •.

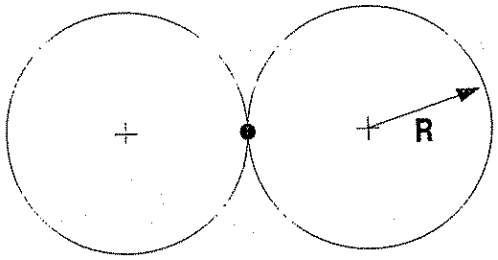


Figure B-21

NOTE: The two arcs can be positioned so that the first arc is inside the second arc, or so that it is outside the second arc. This is shown in Figure B-22.

In Figure B-22, **Arc 2** is shown with a defined radius. The other arc (Arc 1) is shown in two different positions, one labeled **In** the other labeled **Out**. The difference between these is the point at which Arc 1 intersects Arc 2. If the intersection is between the two arc centers, it is called **In**. If the intersection point of the two arcs is not between the two centers, then it is **Out**. This is used to identify which of the two possible arcs is the correct one.

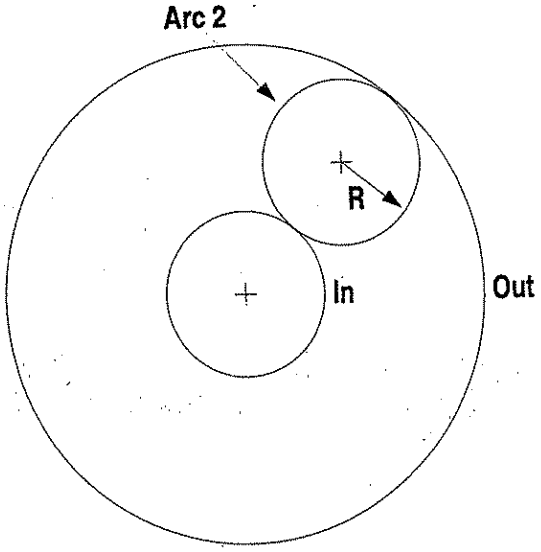


Figure B-22

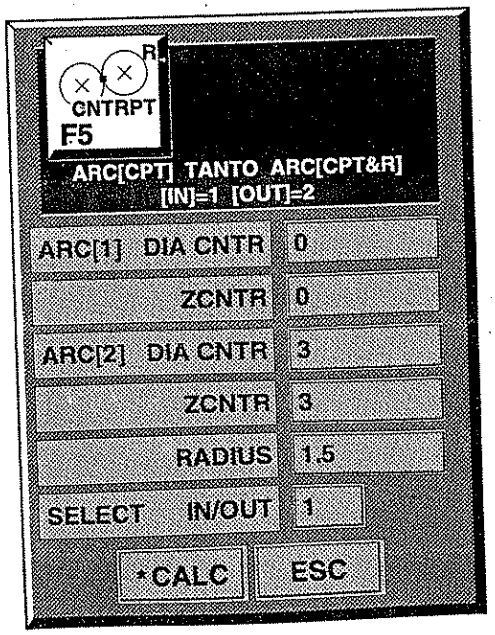


Figure B-23

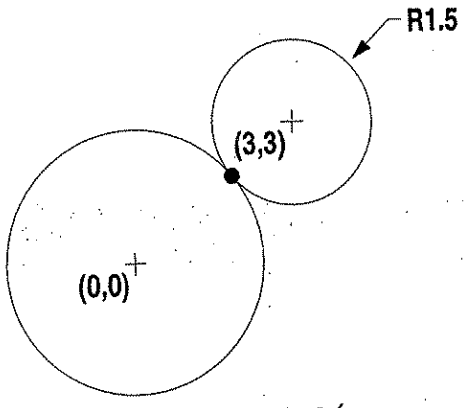


Figure B-24

In the example shown in Figure B-24, the desired arc is **In**. This arc is selected by entering a **1** in the **Select IN/OUT** parameter, shown in Figure B-23.

ARC[1]: **DIA Cntr** This is the X diameter of the arc's centerpoint.
ZCntr This is the Z coordinate of the arc's centerpoint.

ARC[2]: **DIA Cntr** This is the X diameter of the arc's centerpoint.
ZCntr This is the Z coordinate of the arc's centerpoint.
Radius This is the radius value of the second arc.

Select **IN / OUT** This parameter chooses whether the first arc is inside the second arc, or outside.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection of the two arcs, and the radius value of the first arc. The coordinates of the point are entered automatically into the correct data fields of the command.

F6: ARC [R] : THRU PT TANTO LN [THRU 2 PTS]

This command finds the intersection point of an arc, defined by its radius and a pt on the arc, and a tangent line, defined by two points on the line. Figure B-25 shows how the arc and the line are defined. The calculated intersection point is shown with a •.

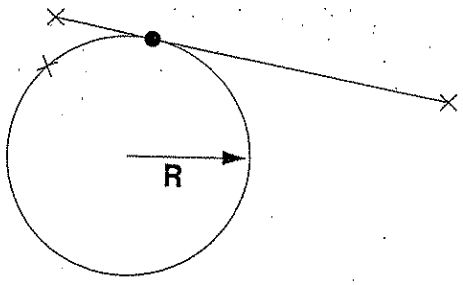


Figure B-25

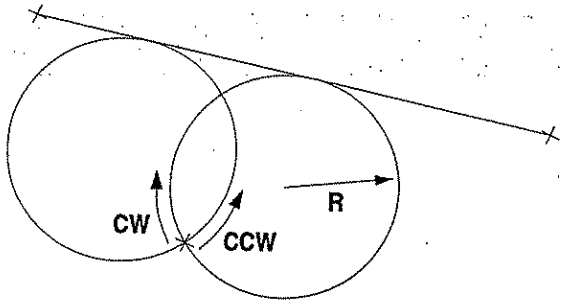


Figure B-26

NOTE: The arc can be positioned so that its direction may be either clockwise or counter-clockwise. This is shown in Figure B-26. The direction is determined by moving from the defined point towards the tangent point of the line and arc.

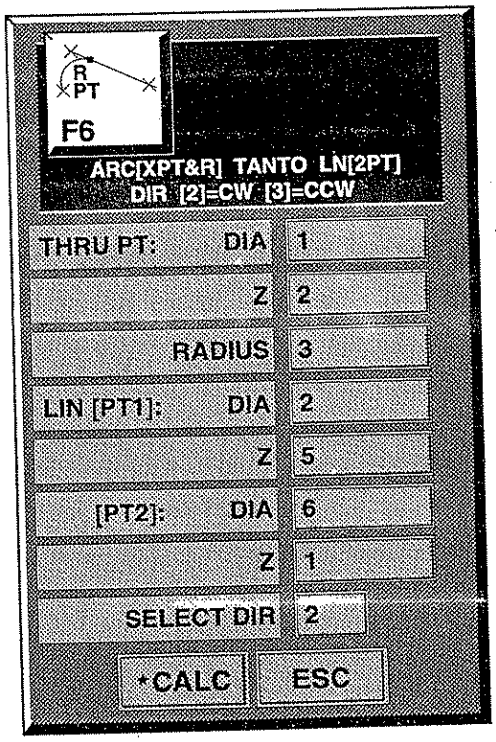


Figure B-27

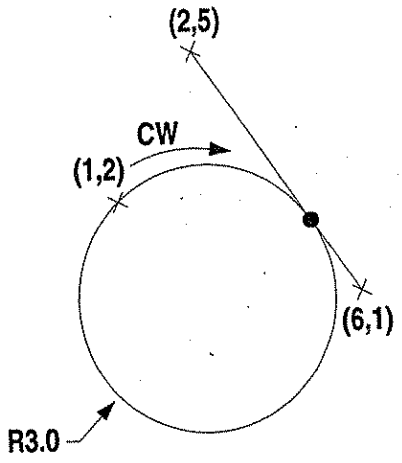


Figure B-28

thru PT:	**DIA**	This is the X diameter of the point on the arc.
	Z	This is the Z coordinate of the point on the arc.
	Radius	This is the radius value of the arc.
LIN [PT1]:	**DIA**	This is the X diameter of the first point on the line.
	Z	This is the Z coordinate of the first point on the line.
LIN [PT2]:	**DIA**	This is the X diameter of the line's second point.
	Z	This is the Z coordinate of the line's second point.
Select	**DIR**	This parameter chooses the direction of the arc, either clockwise, or counter-clockwise.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection of the line and the arc (the arc direction is also shown), and the arc centerpoint. These points are displayed as shown in Figure E-29, so that it one can be selected as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure E-29. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

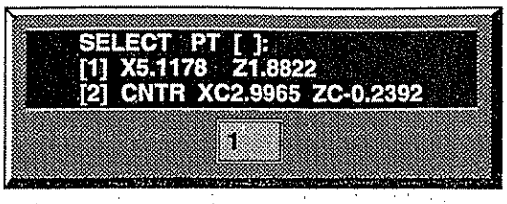


Figure B-29

9: INTOF: 2 LNS [THRU 2 PTS]

This command finds the intersection point of two lines. Each of the two lines is defined by two points. This is shown in Figure B-30.

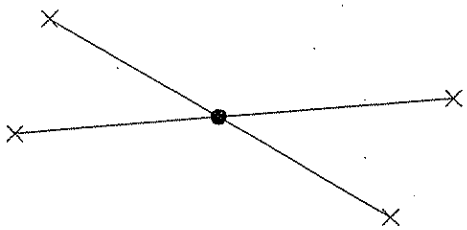


Figure B-30

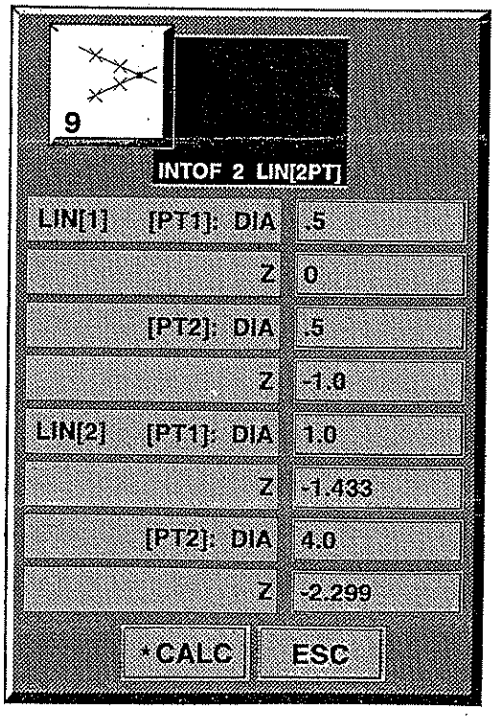


Figure B-31

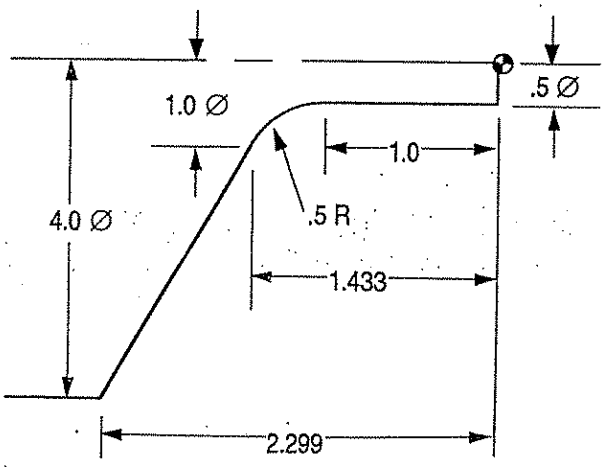


Figure B-32

LIN1 [PT1]:	**DIA**	This is the X diameter of the first point on line 1.
	Z	This is the Z coordinate of the first point on line 1.
LIN1 [PT2]:	**DIA**	This is the X diameter of the second point on line 1.
	Z	This is the Z coordinate of the second point on line 1.
LIN2 [PT1]:	**DIA**	This is the X diameter of the first point on line 2.
	Z	This is the Z coordinate of the first point on line 2.
LIN2 [PT2]:	**DIA**	This is the X diameter of the second point on line 2.
	Z	This is the Z coordinate of the second point on line 2.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection of the two lines. The point is entered automatically into the correct data fields of the command.

8: INTOF: LN [THRU PT] ATANGL TO LN [THRU 2 PTS]

This command finds the intersection point of a line, defined by two points on the line, and a second line, defined by a point on the line, and the angle between the two lines. An example of this is shown in Figure B-33.

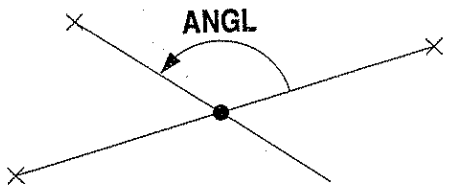


Figure B-33

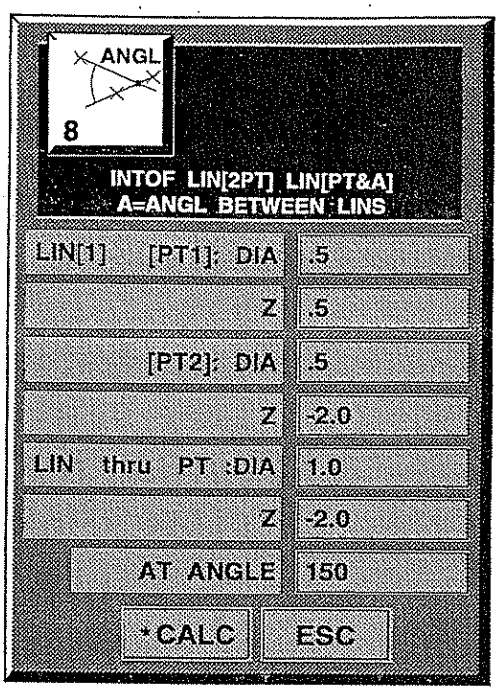


Figure B-34

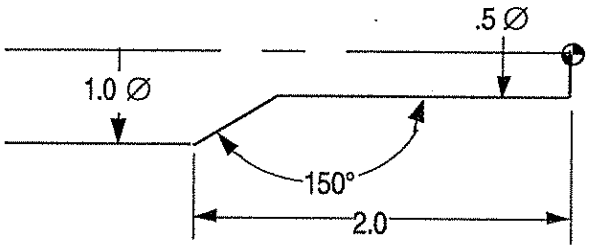


Figure B-35

LIN1 [PT1]: DIA — This is the X diameter of the first point on line 1.
Z — This is the Z coordinate of the first point on line 1.

[PT2]: DIA — This is the X diameter of the second point on line 1.
Z — This is the Z coordinate of the second point on line 1.

LIN thru PT: DIA — This is the X diameter of the point on line 2.
Z — This is the Z coordinate of the point on line 2.

at ANGLE — This is the angle between line 1 and line 2.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection of the two lines. The point is entered automatically into the correct data fields of the command.

7: INTOF: LN [THRU 2 PTS] & ARC [CNTRPT & R]

This command finds the intersection points of a line, defined by two points, and an arc, defined by its centerpoint and radius. See Figure B-36.

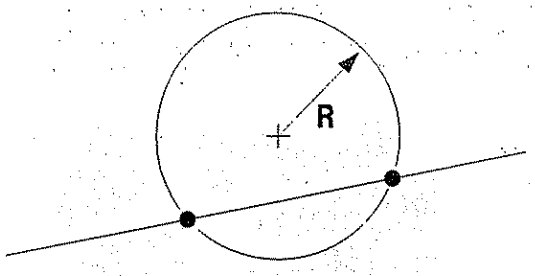


Figure B-36

NOTE: This command calculates two point locations, giving the user a choice between the two. The two calculated points are shown in Figure B-39.

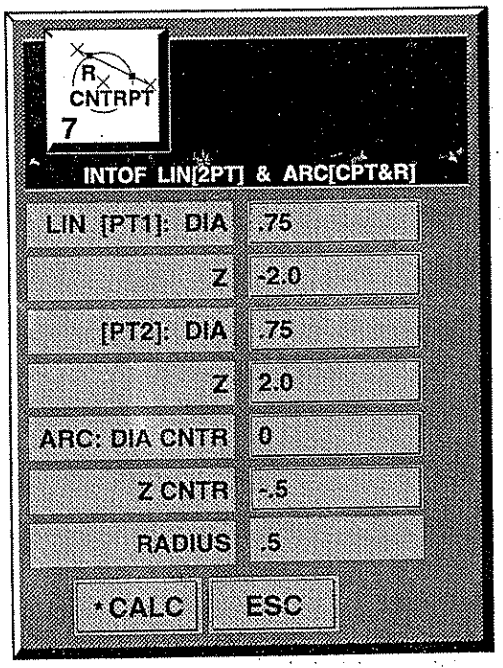


Figure B-37

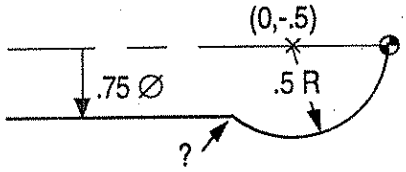


Figure B-38

LIN [PT1]: **DIA** This is the X diameter of the first point of the line.
Z This is the Z coordinate of the first point of the line.

LIN [PT2]: **DIA** This is the X diameter of the second point of the line.
Z This is the Z coordinate of the second point of the line.

ARC: **DIA Cntr** This is the X diameter of the arc's centerpoint.
ZCntr This is the Z coordinate of the arc's centerpoint.

Radius This is the radius value of the arc.

When all of the data is entered, the system calculates the two intersection points for the line and arc. The coordinates of the two points are displayed as shown in Figure B-39. One of these points can be chosen as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure B-39. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

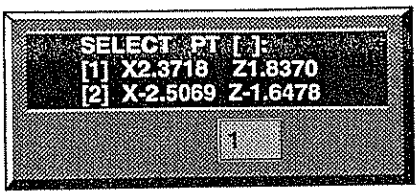


Figure B-39

6: INTOF: LN [THRU 2 PTS] & ARC [THRU 2 PTS & R]

This command calculates the intersection points of a line, defined by 2 points, and an arc, defined by two points on the arc and its radius value. This command returns two intersection points, because the line and arc intersect in more than one location. An example of this is shown in Figure B-40. The user is given a choice of the two points to use as a location in a program or DO EVENT command.

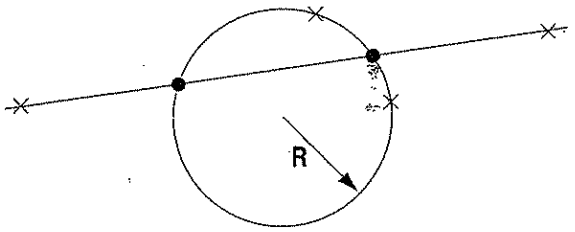


Figure B-40

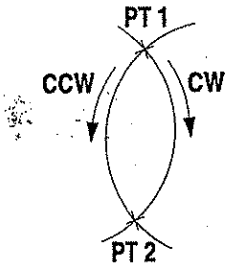


Figure B-41

NOTE: The direction of the arc must be specified as one of the parameters in this command. The direction is determined by moving around the arc from point 1 to point 2. The direction of this movement determines the direction of the arc. An example of this is shown in Figure B-41.

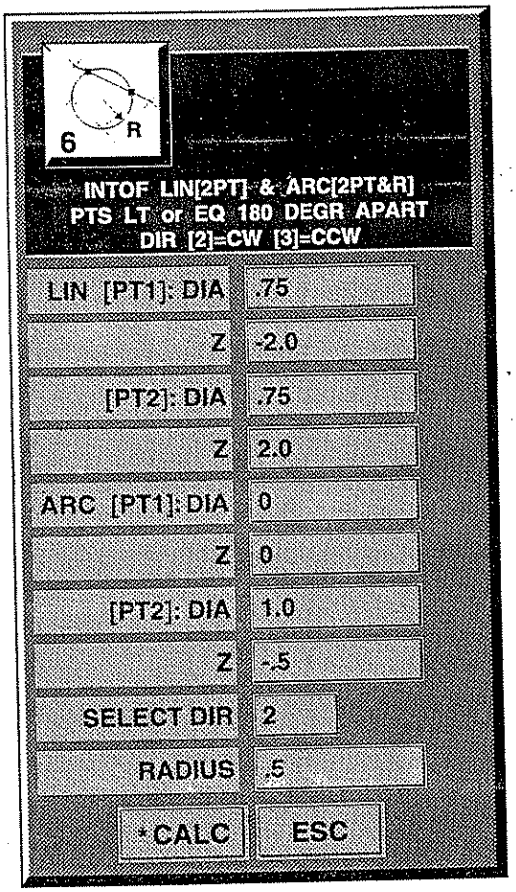


Figure B-42

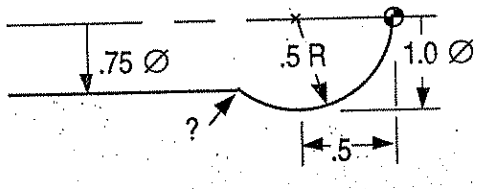


Figure B-43

LIN [PT1]: **DIA** This is the X diameter of the first point of the line.
Z This is the Z coordinate of the first point of the line.

[PT2]: **DIA** This is the X diameter of the second point of the line.
Z This is the Z coordinate of the second point of the line.

ARC: [PT1]: **DIA** This is the X diameter of the first point of the arc.
Z This is the Z coordinate of the first point of the arc.

[PT2]: **DIA** This is the X diameter of the second point of the arc.
Z This is the Z coordinate of the second point of the arc.

Select DIR This is the direction of the arc.

Radius This is the radius of the arc.

When all of the data is entered, the system calculates the two intersection points of the arc and line. The coordinates of the two points are displayed as shown in Figure B-44. One of these points can be chosen as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure B-44. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

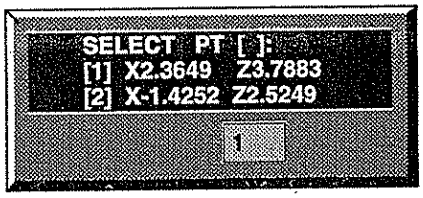


Figure B-44

5: INTOF: 2 ARCS [CNTRPT & R]

This command finds the intersection points of two arcs, each defined by its centerpoint and radius.

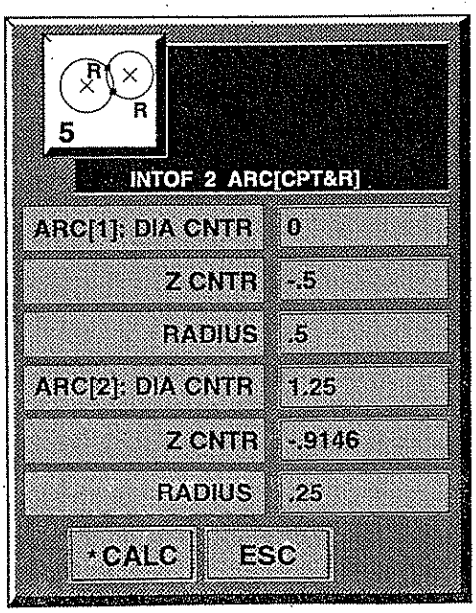


Figure B-46

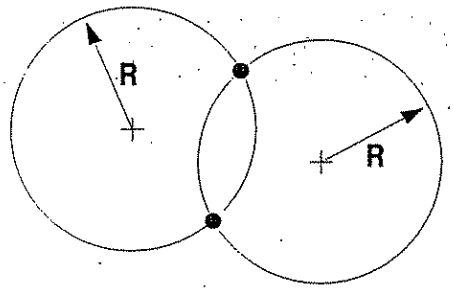


Figure B-45

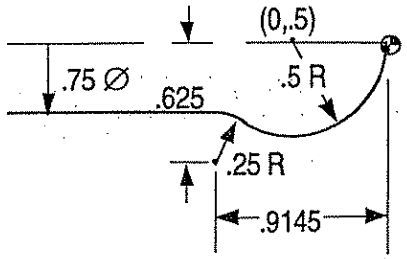


Figure B-47

ARC[1]: **DIA Cntr** This is the X diameter of the first arc's centerpoint.
ZCntr This is the Z coordinate of the first arc's centerpoint.

Radius This is the radius value of the first arc.

ARC[2]: **DIA Cntr** This is the X diameter of the second arc's center.
ZCntr This is the Z coordinate of the second arc's center.

Radius This is the radius value of the second arc.

When all of the data is entered, the system calculates the two intersection points of the two arcs. The coordinates of the two points are displayed as shown in Figure B-48. One of these points can be chosen as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure B-48. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

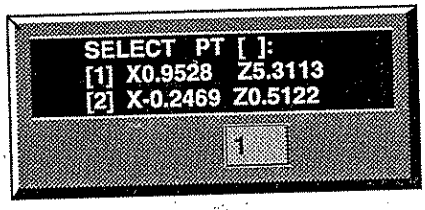


Figure B-48

4: CNTRPT: ARC [THRU 2 PTS w ANGL BETWEEN PTS]

This command calculates the centerpoint location of an arc, defined by two points on the arc, and the angle between them. Figure B-49 shows how the arc is defined.
NOTE: The angle must be less than or equal to 180°.

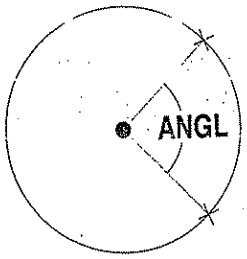


Figure B-49

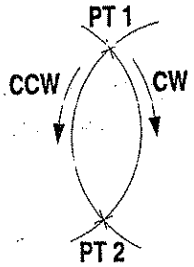


Figure B-50

NOTE: The direction of the arc must be specified as one of the parameters in this command. The direction is determined by moving around the arc from point 1 to point 2. The direction of this movement determines the direction of the arc. An example of this is shown in Figure B-50.

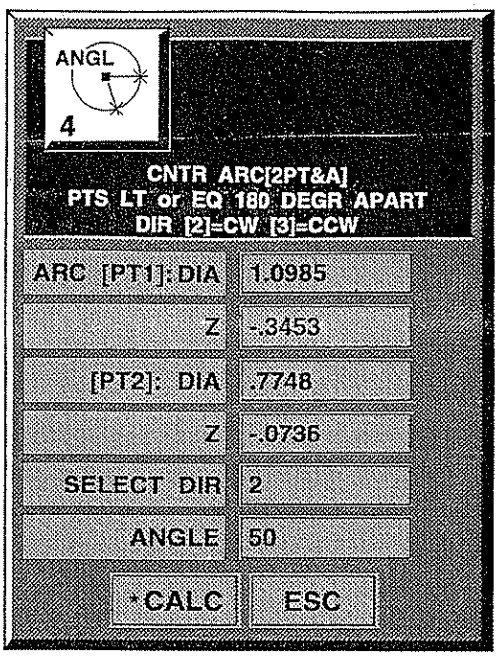


Figure B-51

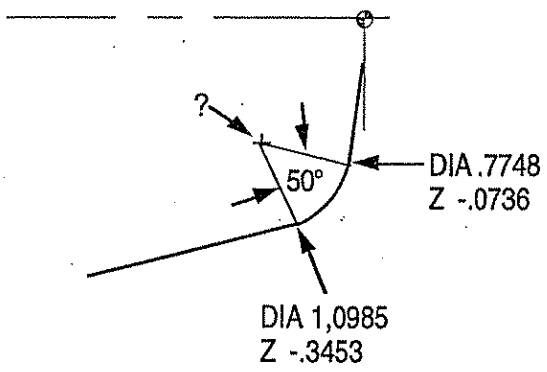


Figure B-52

ARC: [PT1]: **DIA** This is the X diameter of the first point of the arc.
Z This is the Z coordinate of the first point of the arc.

[PT2]: **DIA** This is the X diameter of the second point of the arc.
Z This is the Z coordinate of the second point of the arc.

Select DIR This is the direction of the arc.

ANGLE This is the angle between the two points.

When all of the data is entered, the system calculates the centerpoint of the arc. The coordinates of the point are automatically placed in the correct data fields of the command from which **/GEO** was selected.

3: CNTRPT: ARC [THRU 2 PTS & R]

This command finds the centerpoint location of an arc defined by two points on the arc, and the radius value of the arc. Figure B-53 shows how the arc is defined.
NOTE: The two points on the arc must be less that 180° apart.

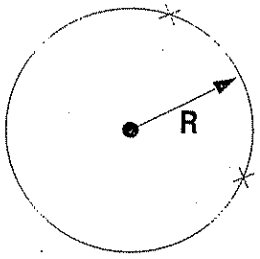


Figure B-53

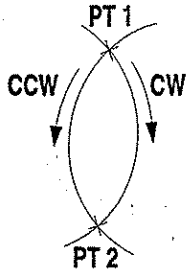


Figure B-54

NOTE: The direction of the arc must be specified as one of the parameters in this command. The direction is determined by moving around the arc from point 1 to point 2. The direction of this movement determines the direction of the arc. An example of this is shown in Figure B-54.

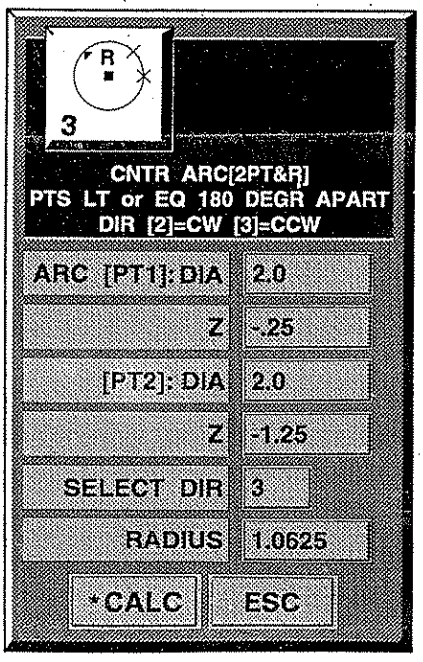


Figure B-55

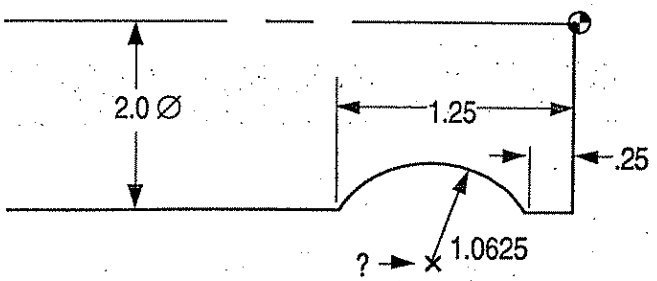


Figure B-56

ARC:	**[PT1]:**	**DIA**	This is the X diameter of the first point of the arc.
		Z	This is the Z coordinate of the first point of the arc.
	[PT2]:	**DIA**	This is the X diameter of the second point of the arc.
		Z	This is the Z coordinate of the second point of the arc.
		Select DIR	This is the direction of the arc.
		Radius	This is the radius of the arc.

When all of the data is entered, the system calculates the centerpoint of the arc. The coordinates of the point are automatically placed in the correct data fields, when the **Enter** key is pressed.

2: CNTRPT: ARC [THRU 3 PTS]

This command finds the centerpoint of an arc defined with three points.

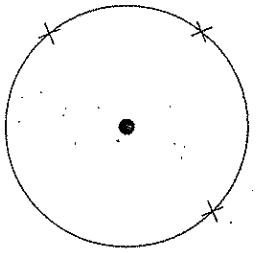


Figure B-57

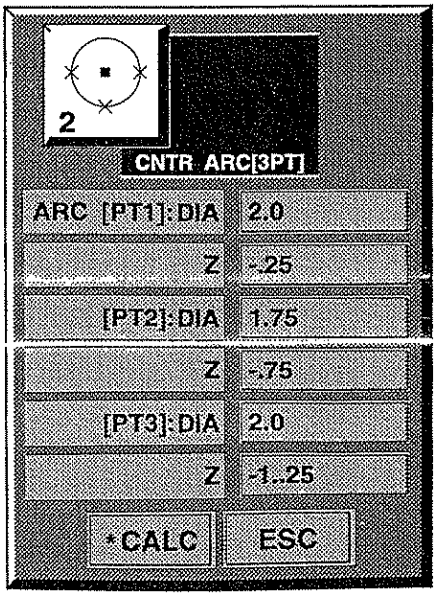


Figure B-58

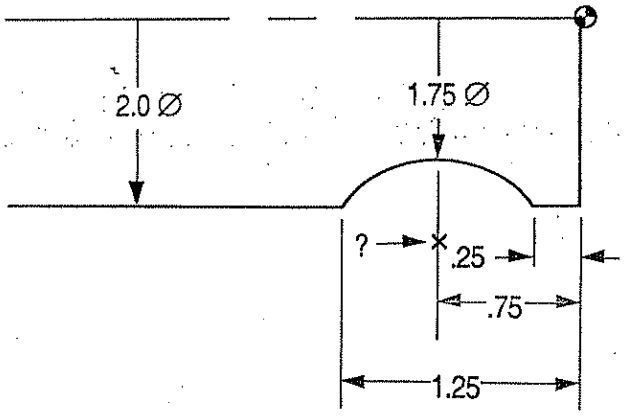


Figure B-59

ARC:	**[PT1]:**	**DIA**	This is the X diameter of the first point of the arc.
		Z	This is the Z coordinate of the first point of the arc.
	[PT2]:	**DIA**	This is the X diameter of the second point of the arc.
		Z	This is the Z coordinate of the second point of the arc.
	[PT3]:	**DIA**	This is the X diameter of the third point of the arc.
		Z	This is the Z coordinate of the third point of the arc.

When all of the data is entered, the system calculates the centerpoint of the arc. The coordinates of the point are automatically placed in the correct data fields, when the **Enter** key is pressed.

1: POLAR: R A DIA C ZC

This command calculates the DIA Z coordinates for a point that is specified using polar coordinates from a specified pole location. Figure B-60 shows how the polar coordinates are defined.

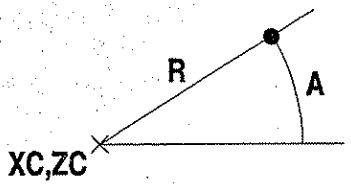


Figure B-60

NOTE: The coordinates of the pole location (DIA C, ZC) must be given in XZ coordinates.

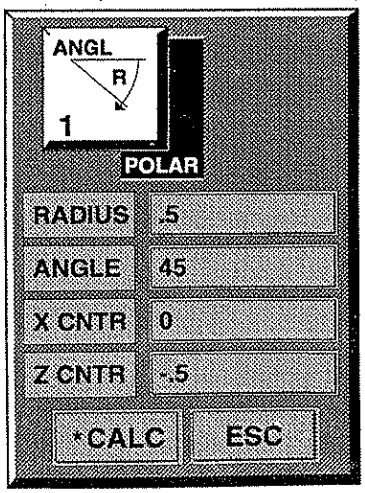


Figure B-61

Angle	This is the angle of rotation from the X axis. This is shown in Figure B-60 as the angle marked **A**.
Radius	This is the radius of the arc.
DIA Cntr	This is the X diameter of the pole location.
ZCntr	This is the Z coordinate of the pole location.

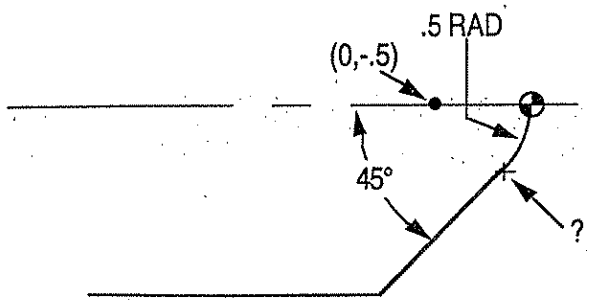


Figure B-62

When all of the data is entered, and **Enter** is pressed, the system calculates the XZ coordinates of the point. The coordinates of the point are entered automatically into the correct data fields of the command.

0: MIDPT| LN[THRU 2PTS]

This command calculates the midpoint of a line which is given by its two endpoints. The coordinates of the two endpoints of the line must be entered, and the EZ-PATH finds the mid-point of the line. Figure B-63, below shows an example of the mid-point command.

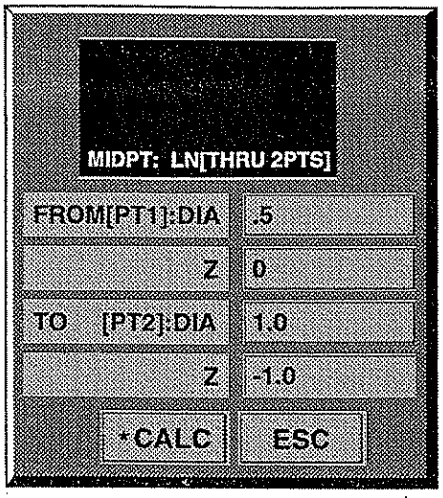


Figure B-63

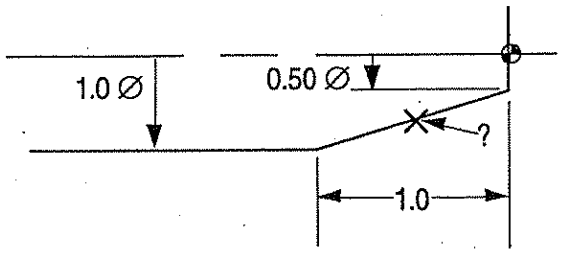


Figure B-64

ESC Pressing the **ESC** key exits the GEOMETRY HELP menu, and returns to the command in which the **/GEO** key was pressed.

Appendix C
Using the Calculator

CALC

In many of the commands throughout the EZ-PATH software, coordinates are required as entered data. Often it is necessary to calculate the correct coordinates using other numbers, or trigonometric functions. These calculations can be carried out by using the **CALC** mode in the EZ-PATH software. This mode is called by pressing the * key on the keyboard whenever coordinates are required.

The screen displays a calculator-like keypad at the bottom of the screen when the **CALC** mode is called. From this screen, complex algebraic equations, and trigonometric functions can be entered and evaluated. When the equation has been entered, press **Enter** to evaluate the equation. The results are shown on the screen in the **CALC** window. Press **Enter** again to place the results in the active coordinate field.

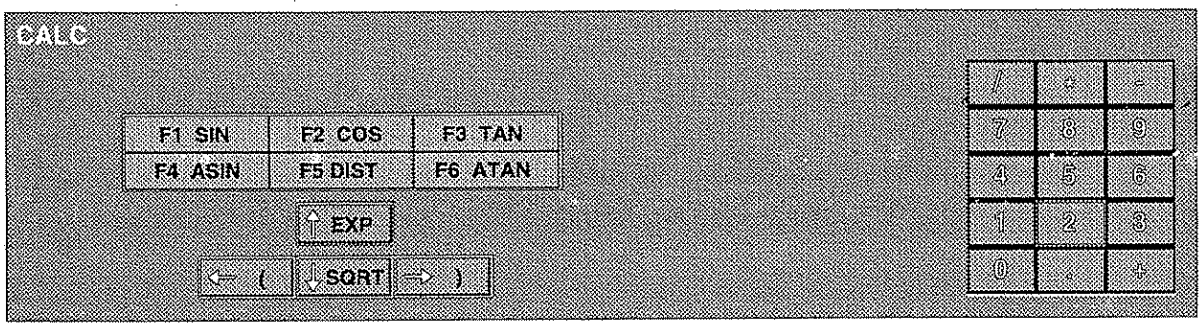


Figure C-1

At the top of the **CALC** window is the word **CALC:**. This shows the equation entered. At the right of the window is the numeric keypad, used for entering numbers and simple mathematical functions like addition, subtraction, division and multiplication using the +, –, /, and * keys.

On the lower left side of the **CALC** window the arrow cursor keys are shown. These are used to enter specific characters. The **<left arrow>** cursor key places an open or left parenthesis into the equation field. Each left parenthesis **(** must be matched by a right parenthesis **)** entered by pressing the **<right arrow>** cursor key.

Complex Functions

When one of the function keys is selected, the screen displays a secondary input field. **[ENTER] FUNC:**

Simple arithmetic functions (+,–,*,/) can be input along with the complex functions listed below.

Some of the complex functions (**ATAN**, and **DIST**) require two numbers to be entered.

F1 SIN This command enters the trigonometric **SIN(** function into the equation field. The operand for this function is a number, positive or negative which represents an angle in a right triangle. The angle is defined in degrees (e.g. 37.5).

F2 COS This command enters the trigonometric **COS(** function into the equation field. The operand for this function is a number, positive or negative which represents an angle in a right triangle. The angle is defined in degrees (e.g. 37.5).

F3 TAN This command enters the trigonometric **TAN(** function into the equation field. The operand for this function is a number, positive or negative which represents an angle in a right triangle. **Note**: Do not enter **90.** degrees for this function. The TAN of 90 degrees is not defined.

F4 ASIN This command enters the **ASIN(** function into the equation field. This function returns the arcsine of the entered number. The number entered for this function must be between **-1.0000** and **1.0000**, otherwise an error is returned by the system, and zero is entered for the coordinate field.

F5 DIST This command enters the **DIST(** function into the equation field. This function requires two numbers. Type the first number, then press **Enter**. Type the second number then press **Enter** again. This function uses the Pythagorean Theorem ($A^2 + B^2 = C^2$) to calculate the length of the third side of a right triangle. The entered numbers are the lengths of the first two sides of the right triangle. The calculated value is the square root of the sum of the squares of the two entered numbers. For example if the entered equation is **DIST(3,4)** then the calculated value is **5.0000**.

F6 ATAN This command enters the trigonometric **ATAN(** function into the equation field. This function requires two numbers separated by a comma, and surrounded by parentheses. The **ATAN** function calculates one angle in a right triangle given the length of the opposite side, and the adjacent side. The first operand should be the length of the side opposite the angle. For example if the entered equation is **ATAN(3,4)** then the calculated value is **36.8966** the measure of the angle opposite the side of length **3**.

<down arrow> SQRT This command enters the **SQRT(** function into the equation field. This function finds the square root of the entered value. If a negative number is entered, the system returns an error and the equation value is set to zero. The number entered in the coordinate field is zero.

<up arrow> EXP This function is used to enter an exponent. The exponent must be a whole number and must not be negative. The system returns the value of the operand if the exponent is either negative or contains a decimal. Fractions also may not be used to express an exponent.

Trigonometric Functions

The six trigonometric functions are derived using a circle with a radius of 1, and a right triangle which lies inside the circle. The relationships of the sides and angles can be calculated using various formulae, as shown in the table and illustration below.

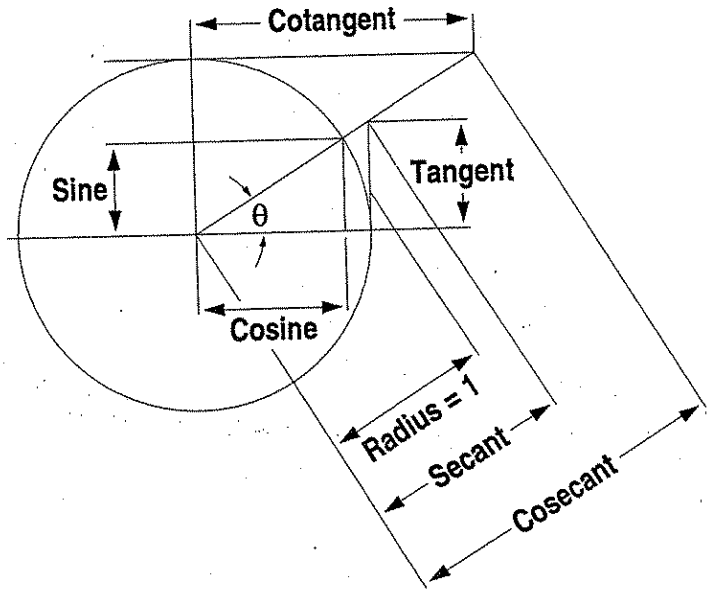


Figure C-2

FORMULAS FOR FINDING FUNCTIONS OF ANGLES
$\frac{\text{Side opposite}}{\text{Hypotenuse}}$ = SINE
$\frac{\text{Side adjacent}}{\text{Hypotenuse}}$ = COSINE
$\frac{\text{Side opposite}}{\text{Side adjacent}}$ = TANGENT
$\frac{\text{Side adjacent}}{\text{Side opposite}}$ = COTANGENT
$\frac{\text{Hypotenuse}}{\text{Side adjacent}}$ = SECANT
$\frac{\text{Hypotenuse}}{\text{Side opposite}}$ = COSECANT
FORMULAS FOR FINDING THE LENGTH OF SIDES FOR RIGHT-ANGLE TRIANGLES WHEN AN ANGLE AND SIDE ARE KNOWN
Length of side opposite { Hypotenuse x Sine / Hypotenuse ÷ Cosecant / Side adjacent x Tangent / Side adjacent ÷ Cotangent
Length of side adjacent { Hypotenuse x Cosine / Hypotenuse ÷ Secant / Side opposite x Cotangent / Side opposite ÷ Tangent
Length of Hypotenuse { Side opposite x Cosecant / Side opposite ÷ Sine / Side adjacent x Secant / Side adjacent ÷ Cosine

Figure C-3

Appendix D
Axes and Coordinates

Some Background Information

When we talk about a lathe cutting a part, we say that the cutter moves in two axes. These are the two directions in which the tool moves in order to cut the material of the part. These directions are forward and backward (the X axis), and left and right (the Z axis). If you watch the lathe in motion, the chuck, and the part itself rotate on the Z axis and the cutting tool, or cutter, moves forward and backward, and left and right.

To make sense of the directions and the values they represent on the screen, stand in front of the machine and look down on the top of it (Figure D-1). When the X coordinate is increasing in value, the motion of the tool is toward the operator. When the Z coordinate is increasing in value, the tool is moving to the operator's right.

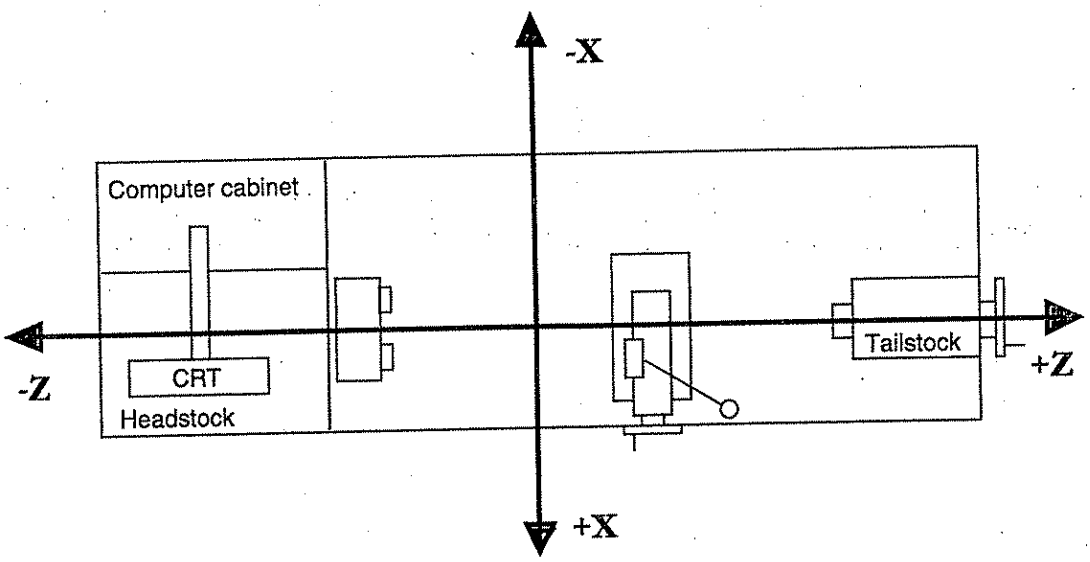


Figure D-1

Axes and Coordinates

The terms **axis** and **coordinate** have already been mentioned, but what do they have to do with the EZ-PATH?

The EZ-PATH (and all CNC machine tools) use a **coordinate system** to identify where the tool is at any one moment and to define where the tool is moving to when it is cutting chips. This coordinate system is called a **Cartesian Coordinate System**.

A Cartesian Coordinate System is based on a grid of lines which have set distances from one central point called the **origin**. Every point on the grid is given both an X and a Z coordinate which show how far to move to get to that point from the origin. Coordinates are usually shown within parentheses **( )** separated by a comma. They are always given in the same order, X first then Z. A Y coordinate is added when the point can move along a third axis, called the Y axis. The Y coordinate on the EZ-PATH is not used since cutting tools on a lathe normally do not move in that direction while in use. The Y coordinate is always listed between the X and Z coordinates when the coordinates are shown in parentheses.

The origin is given the coordinates (0,0) to make things simple. This means that to move to a point labeled **A** that has the coordinates (3,2) (figure D-2), you would start at the origin and move 3 grid lines (usually inches) along the X axis, and then 2 grid lines along the Z axis. Or you could move 2 inches along the Z axis and then 3 inches along the X axis. You will still end up at point **A**, but the path you would take to get there is different.

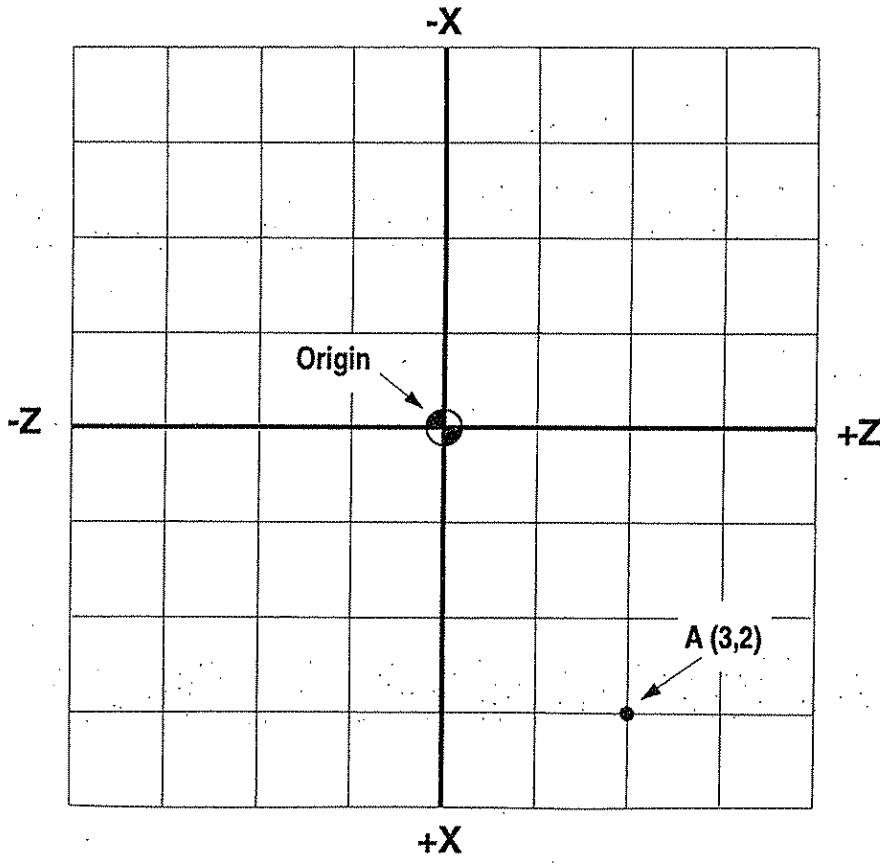


Figure D-2

When you move from one point to another on the grid, you count up or down depending on the direction you are moving to get to the desired point. If you move to the right, you count up, or add to the Z coordinate. If you move left, you count down, or subtract one for each grid line you cross as you move to get to the target point. If you move towards the top of the grid you subtract from the X coordinate, and if you move towards the bottom of the grid, you add one for each line that you cross.

If you move to the left of the X axis, or above the Z axis, you reach zero and then go beyond it. These are negative numbers and are shown with a minus sign, like -1. The minus in -1 shows that you are one line above the Z axis or to the left of the X axis, depending on whether the -1 is the X coordinate or the Z coordinate.

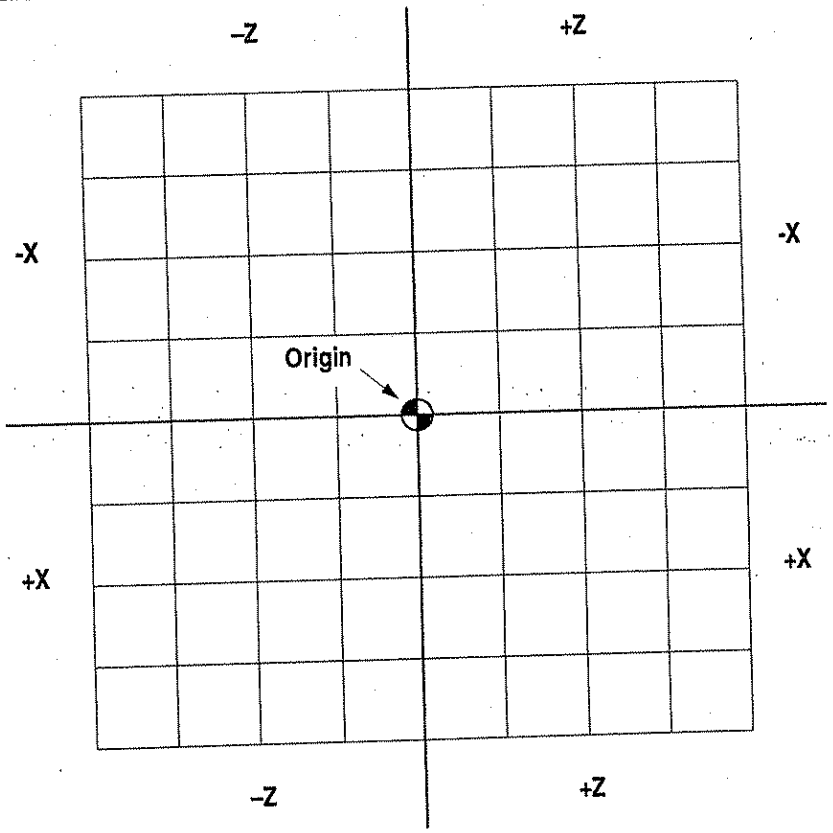


Figure D-3

As you move on the grid, the number of lines that you cross in each direction is the distance that you moved along each axis to get to the new point. This gives two distances for each move, one along the X axis, and one along the Z axis. It is easy to calculate the distances necessary to move from one point to another.

Start with the X coordinate of each point. Take the higher number of the two X coordinates and subtract the lower number. Remember that subtracting a negative number is the same as adding a positive number (e.g. 3 – -2 = 5). The answer is the distance between the X coordinates of the two numbers. Repeat this for the Z coordinates.

For example, in Figure D-4 there are two points, **A** and **B**. The coordinates for **A** are (7,-2) and the coordinates for **B** are (3,5). Take the higher of the two X coordinates and subtract the lower number (7 – 3 = 4). Repeat for the Z coordinates (5 – -2 = 7). The distances are 4, parallel to the X axis, and 7, parallel to the Z axis.

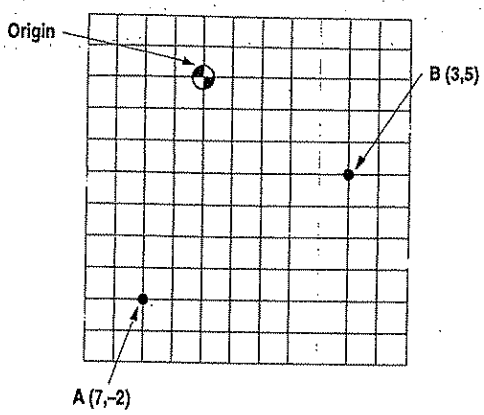


Figure D-4

This does not tell you how to move to get to the new point though. If you were at the point (7,-2) and wanted to move to the point (3,5) you would need to know more than just how far to move. You need to know **what direction** to move.

Compare the X coordinates again. You are starting at a point whose X coordinate is 7, and you want to go to a point whose X coordinate is 3. 7 is greater than 3 so the movement direction is towards the lower numbers at the top of the grid. This gives the direction parallel to the X axis. Now compare the Z coordinates again. You are starting at a point whose Z coordinate is -2. You are moving to a point whose Z coordinate is 5. Since -2 is less than 5, the movement direction is towards the greater numbers on the right of the grid. This gives the direction parallel to the Z axis.

Absolute vs. Incremental Programming

There are two ways to program a machine tool to move from one point to another. You can instruct the machine to move from the current location to the point whose coordinates are (X_1, Z_1). This is called **absolute** programming because the coordinates you give the machine are based on a known location called the origin (0,0).

The second way to program this same move, is to tell the machine tool how far to move in each axis. This means that the point is given as two distances from the current location, one along the X axis, the other along the Z axis. Each number is accepted as a direction and distance. For instance, giving **-2** as the Z distance moves the tool to the left where giving **2** would move the tool to the right. Each of these would move the tool the same distance from the current location, but in opposite directions. This is called **incremental** programming, and it does not matter where the origin is located.

In programming the EZ-PATH, you may use either **absolute** or **incremental** programming. Sometimes it is easier to use one than the other. The program mode is shown in the programmed instructions for the EX-PATH as **ABS** or **INC**. This is usually the third item seen in the instruction line. For example:

```
0000 LINE INC X-4.000 Z7.000 F0.005.
0000 LINE ABS X3.000 Z5.000 F0.005.
```

The first instruction tells the EZ-PATH to move or cut along a linear path at a certain feed rate from the current position, 4 inches in the negative X direction, 7 inches in the positive Z direction, at a feed rate of 0.005 inches per revolution. The second instruction tells the EZ-PATH to move or cut along a linear path at a certain feed rate from the current position to a point 3 inches in the positive X direction and 5 inches in the positive Z direction from the

origin (0,0), at a feed rate of 0.005 inches per revolution.

Below is a grid with some points marked on it. The origin is at the center of the grid. Starting at the first point given, move to the next point, and write down the absolute and incremental values of the move. Remember when you move to the right or towards the bottom of the grid, your movement is positive, when you move left or up, it is negative.

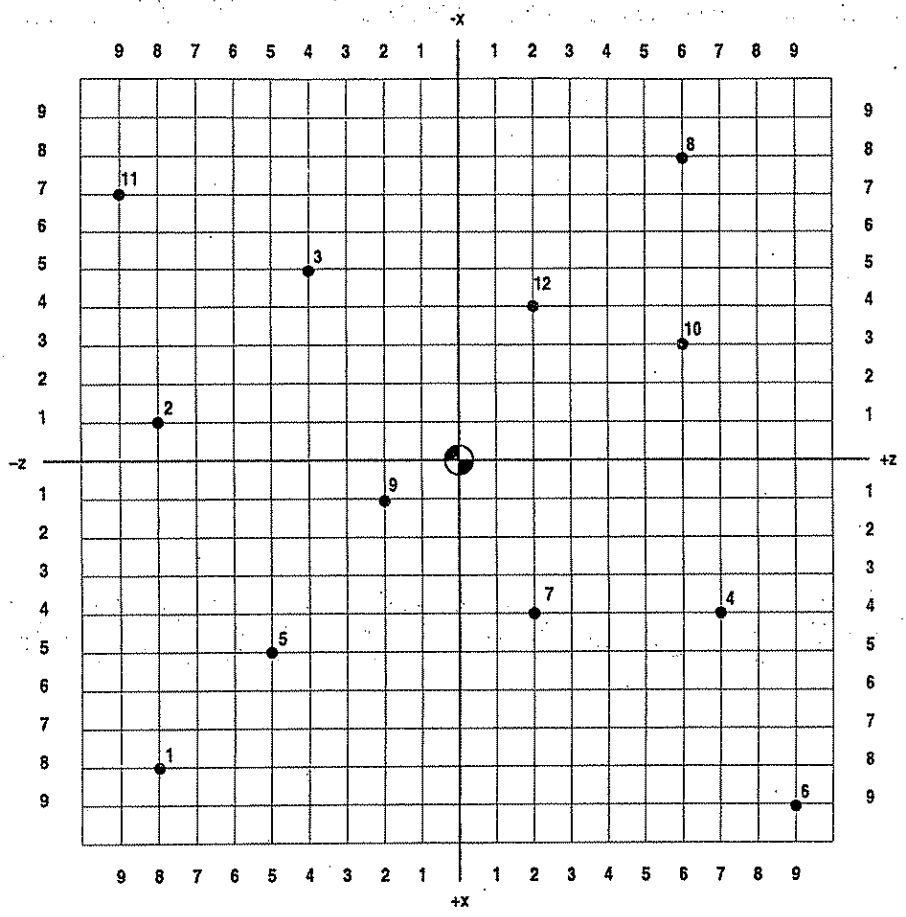


Figure D-5

ABSOLUTE

example:

7 to 11 *X -7* *Z -9*

6 to 1	X_____	Z_____
5 to 4	X_____	Z_____
8 to 3	X_____	Z_____
2 to 4	X_____	Z_____
4 to 5	X_____	Z_____
5 to 6	X_____	Z_____
12 to 7	X_____	Z_____
10 to 8	X_____	Z_____
4 to 9	X_____	Z_____

INCREMENTAL

7 to 11 *X -11* *Z -11*

6 to 1	X_____	Z_____
5 to 4	X_____	Z_____
8 to 3	X_____	Z_____
2 to 4	X_____	Z_____
4 to 5	X_____	Z_____
5 to 6	X_____	Z_____
12 to 7	X_____	Z_____
10 to 8	X_____	Z_____
4 to 9	X_____	Z_____

ANSWERS TO EXERCISES

ABSOLUTE

example:

7 to 11	*X -7*	*Z -9*
6 to 1	X 8	Z -8
5 to 4	X 4	Z 7
8 to 3	X -5	Z -4
2 to 4	X 4	Z 7
4 to 5	X 5	Z -5
5 to 6	X 9	Z 9
12 to 7	X 4	Z 2
10 to 8	X -8	Z 6
4 to 9	X 1	Z -2

INCREMENTAL

7 to 11	*X -11*	*Z -11*
6 to 1	X -1	Z -17
5 to 4	X -1	Z 12
8 to 3	X 3	Z -10
2 to 4	X 5	Z 15
4 to 5	X 1	Z -12
5 to 6	X 4	Z 14
12 to 7	X 8	Z 0
10 to 8	X -5	Z 0
4 to 9	X -3	Z -9

Appendix E
EZ-PATH Programming Example

As an example, the following part is given with the programmed instructions. The blueprint for the part is shown below.

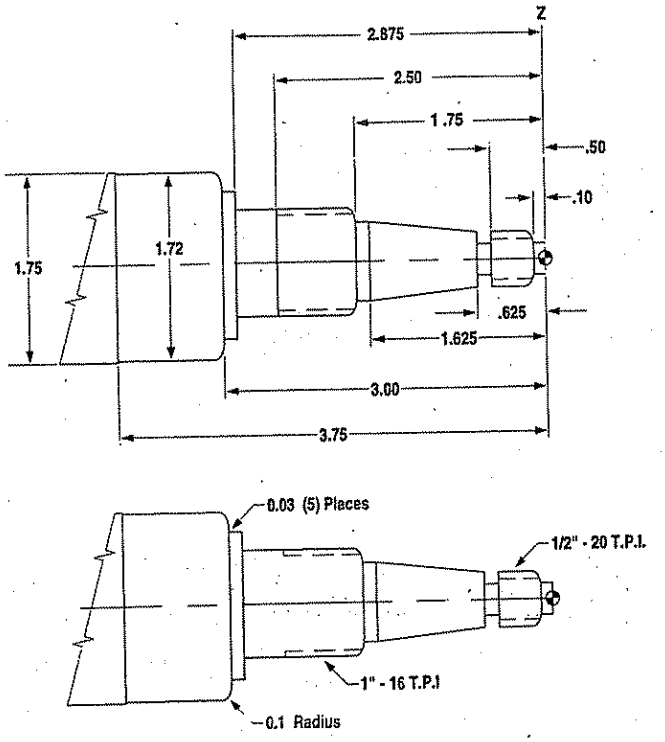


Figure E-1

The tools used to create this part are an OD turning tool, an OD grooving tool and a 60° threading tool. Be sure all the tool offsets are set following the steps outlined in Chapter 3 of this manual.
The suggested material is 1 3/4" diameter aluminum bar. Secure the work material in the spindle so that there is NO LESS THAN 4" hanging out.

To create the program, enter the MDI mode from the Basic Operations screen. The MDI screen will appear and line number 0000 will be created. Do not attempt to delete this line.

The first line of the program is a RAPID move to the absolute coordinates X2.0, Z3.0.

Press the [1] POS key. Enter the following:

DIA ABS 2.0 <ENTER>

Z ABS 3.0 <ENTER>

The program line will appear like this:

0010 RAPID ABS X2.0000 Z3.0000

The second line of the program should be a tool change. This will create a screen prompt so that the operator has an opportunity to load the correct tool for the start of the program. This is essential if the program will be repeated and it requires more than one tool.

Press the [7] TLCHG key. Enter the correct ID number, Tool number and Offset number that are set for the turning tool.

The program line will appear similar to this:

0020 TOOLCHG I1 T01 01

The next line of the program gives the operator a screen prompt to manually change the spindle speed.

Press the [6] SETRPM key. Enter the following:

SET RPM 1060 <ENTER>

The program line will appear like this:

0030 SETRPM S1060

The next line of program is the STARTPATH command. This command instructs the machine to create a path with the following commands that will be executed with a roughing command.

Press the [F1] PATH key. Enter the following:

PATH NAME 1 <ENTER>

The program line will appear like this:

0040 STARTPATH 1

NOTE: In the PATH mode only a few functions are available. They are:

[1] POS [2] LINE [3] ARC [—] UNDO

[4] BL LIN [5] BL ARC [6] PTHSTP

The next program line is a RAPID command. It instructs the machine to move to a specified point.

Press the [1] POS key. Enter the following:

DIA ABS 0.0 <ENTER>

Z ABS 0.05 <ENTER>

The program line will appear like this:

0050 RAPID ABS X0.0000 Z0.0500

The following program lines create the small diameter end of the part.

Press the [2] LINE key. Enter the following:

DIA ABS 0.0 <ENTER>

Z ABS 0.0 ENTER

Press the [2] LINE key again. Enter the following:

DIA ABS 0.3 <ENTER>

Z ABS -0.0 <ENTER>

Press the [2] LINE key again. Enter the following:

DIA ABS 0.3 <ENTER>

Z ABS -0.1 <ENTER>

The three lines of the program will look like this:

0060 LINE ABS X0.0000 Z0.0000 F0.0050
0070 LINE ABS X0.3000 Z0.0000 F0.0050
0080 LINE ABS X0.3000 Z-0.1000 F0.0050

NOTE: while creating a path, the feed rate is automatically input with a default value.

This next command creates the chamfer at the beginning of the thread on the small end of the part.

Press the [4] BL LIN key. Enter the following:
DIA ABS 0.5 <ENTER>
Z ABS -0.1 <ENTER>
CHAMF/R BLEND 0.03 <ENTER>
CHAMF/CW/CCW 1 <ENTER>

The program line will look like this:

0090 CHAMFER ABS X0.5000 Z-0.1000 P0.0300 P0.0300 F0.0050

The next program line creates the small threaded section diameter .

Press the [2] LINE key. Enter the following:
DIA ABS 0.5 <ENTER>
Z ABS -0.625 <ENTER>

The program line will look like this:

0100 LINE ABS X0.5000 Z-0.625 F0.0050

This LINE command creates the taper on the part.

Press the [2] LINE key. Enter the following:
DIA ABS 0.75 <ENTER>
Z ABS -1.625 <ENTER>

The program line will look like this:

0110 LINE ABS X0.7500 Z-1.625 F0.0050

The next line of the program creates the short flat segment at the large end of the taper.

Press the [2] LINE key. Enter the following:
DIA ABS 0.75 <ENTER>
Z ABS -1.75 <ENTER>

The program line will look like this:

0120 LINE ABS X0.7500 Z-1.7500 F0.0050

The next program line creates the chamfer at the beginning of the large threaded section.

Press the [4] BL LINE key. Enter the following:
DIA ABS 1.0 <ENTER>
Z ABS -1.75 <ENTER>
CHAMF/R BLEND 0.03 <ENTER>
CHAMF/CW/CCW 1 <ENTER>

The program line will look like this:

0130 CHAMFER ABS X1.0000 Z-1.7500 P0.0300 P0.0300 F0.0050

The next command creates the large threaded section diameter.

Press the [2] LINE key. Enter the following:
DIA ABS 1.0 <ENTER>
Z ABS -2.875 <ENTER>
The program line will look like this:
0140 LINE ABS X1.0000 Z-2.8750 F0.0050

The next line of the program creates a small chamfer on the second largest diameter of the part.

Press the [4] BL LINE key. Enter the following:
DIA ABS 1.375 <ENTER>
Z ABS -2.875 <ENTER>
CHAMF/R BLEND 0.03 <ENTER>
CHAMF/CW/CCW 1 <ENTER>
The program line will look like this:
0150 CHAMFER ABS X1.3750 Z-2.8750 P0.0300 P0.0300 F0.0050

This LINE command creates another short flat segment on the part.

Press the [2] LINE key. Enter the following:
DIA ABS 1.375 <ENTER>
Z ABS -3.0 <ENTER>
The program line will look like this:
0160 LINE ABS X1.3750 Z-3.0000

The next command creates the rounded edge of the largest diameter of the part.

Press the [4] BL LINE key. Enter the following:
DIA ABS 1.72 <ENTER>
Z ABS -3.00 <ENTER>
CHAMF/R BLEND 0.1 <ENTER>
CHAMF/CW/CCW 2 <ENTER>
The program line will look like this:
0170 BLEND I LN ABS X1.7200 Z-3.0000 R0.1000 CW F0.0050

The next line of the program creates the final flat segment of the part.

Press the [2] LINE key. Enter the following:
DIA ABS 1.72 <ENTER>
Z ABS -3.75 <ENTER>
The program line will look like this:
0180 LINE ABS X1.72000 Z-3.7500 F0.0050

The next line of the program moves the tool away from the final part diameter.

Press the [2] LINE key. Enter the following:
DIA ABS 2.0 <ENTER>
Z ABS -3.75 <ENTER>
The program line will look like this:
0190 LINE ABS X2.0000 Z-3.7500 F0.0050

The last line of the PATH is a PATHSTOP. This line is necessary to let the machine know that the profile of the part is complete.

Press the [6] PTHSTP command. Enter a **0 <ENTER>** to confirm the PATHSTOP.
The program line will look like this:

0200 PATHSTOP

The next command instructs the machine to rough out all the unnecessary material to make the final part based on the PATH that was entered.

Press the [F2] ROUGH key. Enter the following:

PATH ID 1 <ENTER>
1-OD/2-ID/3-FACE 1 <ENTER>
X FIN ALLOW 0.01 <ENTER>
Z FIN ALLOW 0.01 <ENTER>
ENGAGE FEED 0.01 <ENTER>
CROSS FEED 0.01 <ENTER>
RETRACT FEED 0.01 <ENTER>
CUT STEP 0.1 <ENTER>
CLEARANCE 0.1 <ENTER>
WITHDRAW ANG 45.0 <ENTER>
WITHDRAW LEN 0.05 <ENTER>
CUT DIR 1-POS/2-NEG 2 <ENTER>
UNDERCUT 1-ON/2-OFF 1 <ENTER>
AUTO ROUND 1-ON/2-OFF 1 <ENTER>

The program line will look like this:

0210 ROUGH 1 I1 X0.0100 Z0.0100 F0.0100 0.0100 0.0100 S0.1000 C0.10000
W45.0000 W0.0500 D2 U1 A1

The next command will instruct the machine to move the tool away from the part.

Press the [1] POS key. Enter the following:

DIA ABS 2.0 <ENTER>
Z ABS 0.1 <ENTER>

The program line will appear like this:

0220 RAPID ABS X2.0000 Z0.1000

This SETRPM command pauses the program and gives the operator a screen prompt to manually change the spindle speed.

Press the [6] SETRPM key. Enter the following:

SET RPM 2120 <ENTER>

The program line will appear like this:

0230 SETRPM S2120

The next command is a PROFILE command. It uses a specified path to put a finish cut on the part.

Press the [F3] PROFIL key. Enter the following:
PATH ID 1 <ENTER>
TOOL POS 1-LEFT/2-RIGHT 1 <ENTER>
X FIN ALLOW 0.0 <ENTER>
Z FIN ALLOW 0.0 <ENTER>
FEED RATE 0.01 <ENTER>
CLEARANCE 0.1 <ENTER>
ENGAGE ANG 45.0 <ENTER>
WITHDRAW ANG 135.0 <ENTER>
UNDERCUT 1-ON/2-OFF 1 <ENTER>
AUTO ROUND 1-ON/2-OFF 1 <ENTER>

The program line will appear like this:
0240 PROFIL 1 X0.0000 Z0.0000 F0.01000 E45.0000 W135.0000 U1 A1

This RAPID command instructs the machine to move the tool away from the work piece.

Press the [1] POS key. Enter the following:
DIA ABS 2.0 <ENTER>
Z ABS 3.0 <ENTER>

The program line will appear like this:
0250 RAPID ABS X2.0000 Z3.0000

The next command gives a screen prompt for the operator to manually change the spindle speed.

Press the [6] SETRPM key. Enter the following:
SET RPM 1060 <ENTER>

The program line will appear like this:
0260 SETRPM S1060

The next command gives a screen prompt for the operator to manually change the tool.

Press the [7] TLCHG key. Enter the correct ID number, Tool number and Offset number that are set for the grooving tool.

The program line will appear similar to this:
0270 TLCHG I93 T03 03

These next two commands instruct the machine to move the tool in position to make the groove on the part.

Press the [1] POS key. Enter the following:
DIA ABS 3.0 <ENTER>
Z ABS -0.625 <ENTER>

Press the [1] POS key again. Enter the following:
DIA ABS 0.6 <ENTER>
Z ABS -0.625 <ENTER>

The program lines will appear like this:
0280 RAPID ABS X3.0000 Z-0.6250
0290 RAPID ABS X0.6000 Z-0.6250

This next LINE command instructs the machine to cut the groove on the part.

Press the [2] LINE key. Enter the following:

DIA ABS 0.3 <ENTER>
Z ABS -0.625 <ENTER>
F FEED 0.005 <ENTER>

The program line will look like this:

0300 LINE ABS X0.3000 Z-0.6250 F0.0050

The DWELL command instructs the machine to hold a tool in a previously specified position for a period of time.

Press the 8 DWELL key. Enter the following:

SECONDS 0.5 <ENTER>

The program line will look like this:

0310 DWELL S0.5000

The next line of the program instructs the machine to retract the tool from the groove.

Press the [1] POS key. Enter the following:

DIA ABS 3.0 <ENTER>
Z ABS -0.625 <ENTER>

The program lines will appear like this:

0320 RAPID ABS X3.0000 Z-0.6250

The next RAPID command instructs the machine to move the tool away from the work piece.

Press the [1] POS key. Enter the following:

DIA ABS 3.0 <ENTER>
Z ABS 3.0 <ENTER>

The program lines will appear like this:

0330 RAPID ABS X3.0000 Z3.0000

The next command gives a screen prompt for the operator to manually change the tool.

Press the [7] TLCHG key. Enter the correct ID number, Tool number and Offset number that are set for the threading tool.

The program line will appear similar to this:

0340 TLCHG I60 T05 05

The next command gives a screen prompt for the operator to manually change the spindle speed.

Press the [6] SETRPM key. Enter the following:

SET RPM 530 <ENTER>

The program line will appear like this:

0350 SETRPM S530

The next line of the program creates the thread on the small diameter of the part.

Press the [F5] THREAD key. Enter the following:

1=OD/2=ID 1 <ENTER>
LEAD 0.0769 <ENTER>
THREAD HEIGHT 0.0472 <ENTER>

NOTE: thread height is an automatically calculated value

STEP 1 0.005 <ENTER>
STEP 2 0.003 <ENTER>
MIN STEP 0.002 <ENTER>
 # SPRING PASSES 2 <ENTER>
WITHDRAW CLEARANCE 0.1 <ENTER>
ENGAGE ANGLE 29 <ENTER>
START Z 0.15 <ENTER>
END Z -0.55 <ENTER>
START DIA 0.5 <ENTER>
END DIA 0.5 <ENTER>

The program line will appear like this:

0360 THREAD 1 L0.0769 H0.0472 S0.0050 0.0030 0.0020 #2 C0.1000 Z0.1538
-0.5500 D0.5000 0.5000

These following commands instruct the machine to withdraw from the first threading operation and position the tool for the second threading operation.

Press the [1] POS key. Enter the following:

DIA ABS 1.0 <ENTER>
Z ABS 0.05 <ENTER>

Press the [1] POS key again. Enter the following:

DIA ABS 1.0 <ENTER>
Z ABS -1.65 <ENTER>

The program lines will appear like this:

0370 RAPID ABS X1.0000 Z0.0500
0380 RAPID ABS X1.0000 Z-1.6500

This THREAD command creates the threads on the large section of the part.

Press the [F5] THREAD key. Enter the following:

1=OD/2=ID 1 <ENTER>
LEAD 0.0625 <ENTER>
THREAD HEIGHT 0.0383 <ENTER>

NOTE: thread height is an automatically calculated value

STEP 1 0.005 <ENTER>
STEP 2 0.003 <ENTER>
MIN STEP 0.002 <ENTER>
 # SPRING PASSES 2 <ENTER>
WITHDRAW CLEARANCE 0.1 <ENTER>
ENGAGE ANGLE 29 <ENTER>
START Z -1.65 <ENTER>

END Z -2.5 <ENTER>
START DIA 1.0 <ENTER>
END DIA 1.0 <ENTER>

The program line will appear like this:

0390 THREAD 1 L0.0625 H0.0383 S0.0050 0.0030 0.0020 #2 C0.1000 Z-1.6500
-2.5000 D1.0000 1.0000

These following commands instruct the machine to withdraw from the threading operation and position the tool away from the work piece.

Press the [1] POS key. Enter the following:

DIA ABS 3.0 <ENTER>
Z ABS -1.65 <ENTER>

Press the [1] POS key again. Enter the following:

DIA ABS 3.0 <ENTER>
Z ABS 3.0 <ENTER>

The program lines will appear like this:

0400 RAPID ABS X3.0000 Z-1.6500
0410 RAPID ABS X3.0000 Z3.0000

The next line of the program gives a screen prompt for the operator to manually stop the spindle.

Press the [.] AUXFUN key. Enter the following:

PRGM 5 <ENTER>

The program line will appear like this:

0420 AUXFUN M5

The last line of the program will reset the program so it can be immediately run again.

Press the [.] AUXFUN key. Enter the following:

PRGM 2 <ENTER>

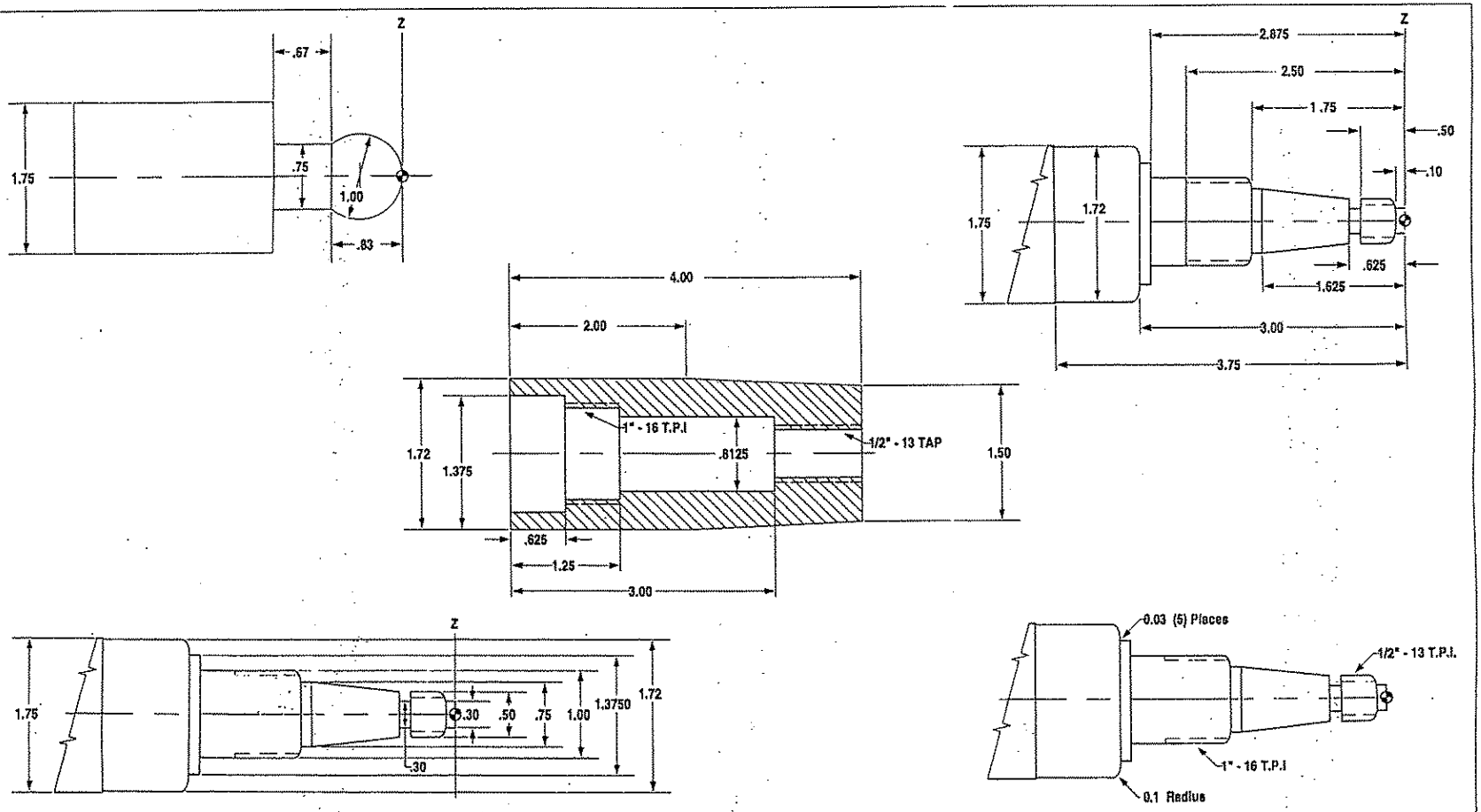
The program line will appear like this:

0430 AUXFUN M2

The program is now completely written. To save the program, press the [0] EXIT key. Enter the name of the program and then select one of the following:

[+] SAVE
[*] SAVE | RUN
[/] SAVE | VIEW
[-] CANCEL

The following page has a blueprint with other parts that can be programmed for the EZ-PATH.



NOTES APPLY UNLESS OTHERWISE SPECIFIED:
1. FILLET RADII MAX
2. √ TO BE SURF. FINISH.
3. BREAK ALL SHARP EDGES. TO .015R MAX.
4. ALL THREADS CLASS 2.
5. MACHINING TOLERANCES:
.X = 1.065
.XX = 1.015
.XXX = 1.005
.XXXX = 1.0005

THIRD ANGLE PROJECTION
INCH

SAND CASTING TOLERANCES:
0"-12" = 1.03
13"-24" = 1.06
OVER 24" = 1.09

MATL.

HT. TR.

GEOMETRIC TOLERANCE UNLESS OTHERWISE NOTED		
⌯	SYMMETRY	.005
⊕	TRUE POSITION	.010
▱	FLATNESS	.005
⊥	PERPENDICULARITY	.005
//	PARALLELISM	.005
∠	ANGULARITY	.005
↗	RUNOUT	.004
⌓	PROFILE OF A SURF.	.003
○	ROUNDNESS	.002
—	STRAIGHTNESS	.005
◎	CONCENTRICITY	.002
⌭	CYLINDERICITY	.002

NEXT ASSY
DRN
DATE
CK'D
DATE
APP'D
DATE

Bridgeport®
BRIDGEPORT MACHINES, INC.

TITLE
PATH DEMO PART

SIZE	FSCM NO.	DWG. NO.	REV.
A	08607	PATH - 1	

SCALE | RELEASE DATE | SHEET OF

Index

A

B

C

D

E-F

G-H

I-K

L-M

N-P

Q-R

S

T

U-W

X-Z