Introduction

The Romi EZPATH® II lathe is part of the Bridgeport Machines PCNC (PC-based Numeric Control) product line, following in the footsteps of such products as the EZTRAK mill and the EZPATH I lathe. It is specifically designed for the first-time PCNC user and is intended to bridge the gap between a handwheel-driven lathe and a full-featured CNC machine, combining the ease of use of one with the increased productivity of the other. Like all machines in the Bridgeport EZ series, it gives the user the flexibility of three different machines in one package:

- In the manual mode, the operator can use the the axis handwheels and advanced digital readout in the same way that a conventional lathe is operated.
- The semi-automatic Do Event mode lets the operator perform complex operations with a single instruction—including tapering, chamfering, and radius cuts.
- In the fully automatic mode, the EZPATH can run complete part programs. In addition to running existing programs, the EZPATH comes with powerful, easy-to-use tools to create new programs. MDI (manual data input) mode is a step-by-step conversational environment which prompts the operator for all the information required to easily machine complex parts. In Save Point (Teach) mode, the operator can manually turn the first part and save the coordinates of each move to replay the operations for subsequent parts. EZPATH also incorporates powerful canned cycles and cutter compensation routines, giving it the ability to machine parts that previously could only be produced on high-end lathes.

The EZPATH requires no prior knowledge of CNC programming or computer experience. By following on-screen prompts, the operator can begin turning a part after only a few minutes of basic training. The programming environment in the EZPATH intelligently prompts the user for basic part information found directly on a blueprint, and even provides math help functions for calculating necessary points.

This manual provides the information to operate and program the EZPATH II. It includes a complete step-by-step tutorial in which the user produces an actual part, a programming example, and a comprehensive index.

Safety Instructions— READ THIS FIRST!

Like most manufacturers, we go to great lengths to make our products as safe as possible. But operators can still get hurt. In virtually every case, the injury is the result of:

- Not knowing how to operate the machine properly;
- Not following proper operating and safety procedures;
- Carelessness or inattention;
- Trying to "take a short cut;"
- Poor maintenance.

For your personal safety, and to get the maximum efficiency out of this precision machine, read and follow operation and maintenance instructions carefully.

- Some parts in the electric and control panels, transformers, motors, junction boxes and other components use high voltage. If these points or terminals are touched they may cause serious electric shock and even death.
- Never touch a manual control (buttons, keys, switches, levers, etc.) with wet hands, shoes, or clothes. This may cause electric shock and even death.
- Warm up the machine—especially the main spindle—by operating it for 10–20 minutes at 1/2 or 1/3 maximum speed.
- Tooling should be in accordance with the machine specifications, size, and type.
- Worn-out tools may cause damage. Replace worn-out tools before damage occurs.
- The work area should be well-illuminated to facilitate safety inspection.
- Tools and other items should be stored. To prevent accidents, do not leave tools around the machine. Tools should not be put on top of the headstock, covers, or similar places. Keep aisles clean.
- To avoid interference, tool length should be within tolerances.

- After mounting a tool, make a test.
- After machining chuck jaws, make sure they hold the workpiece with proper clamping force.
- Do not work with long loose hair that may get tangled with the machine; tie on top of the head or to the back.
- Do not operate manual controls (buttons, keys, switches, levers, etc.) with gloves; this may cause defects.
- Whenever a heavy workpiece, part, or component must be moved, and whenever there is any risk, two or more people should work together.
- Only trained and qualified workers should operate forklift trucks, cranes, and similar equipment to avoid collision and damage.
- Hold workpieces and parts firmly and securely.
- Do not touch chips or tool tip with unprotected hands.
- Stop the machine before adjusting cooling system nozzle.
- Never touch with hands or in any other way a rotating workpiece or main spindle.
- Do not open the chuck guards, rear splash guard or machine doors during machining operation.
- Use a brush to remove chips from the tool tip—NEVER WITH UNPROTECTED HANDS.
- Stop the machine whenever mounting or removing a tool.
- Wear a mask or face protection whenever machining a magnesium alloy workpiece.
- During heavy machining, avoid chip accumulation. Hot chips may cause a fire.

Other important safety information is contained in the *EZPATH II Maintenance & Installation Manual*. Make sure you understand *all* safety procedures before operating the machine.

Contents

Chapter 1/Overview

Chapter 2/Basic Operations

Chapter 3/Using Tools

Chapter 4/Tutorial

Chapter 5/Do Event mode

Chapter 6/Creating Programs

Chapter 7/Editing Programs

Chapter 8/Running Programs

Chapter 9/File Utilities

Chapter 10/Help

Index

Programming Workbook

Chapter 1
Overview

Figure 1-1 illustrates the main operating features of the EZPATH II lathe. This chapter will describe each of the operating controls located on the machine, as well as start-up procedures. Throughout this manual, it is assumed that you are familiar with basic turning principles and procedures; the manual focuses instead on how to apply the EZPATH controls and features to accomplishing those tasks. For more information on the initial set-up and configuration of the machine, consult the *EZPATH II Installation and Maintenance Manual,* and the *EZPATH II Electrical Manual.*

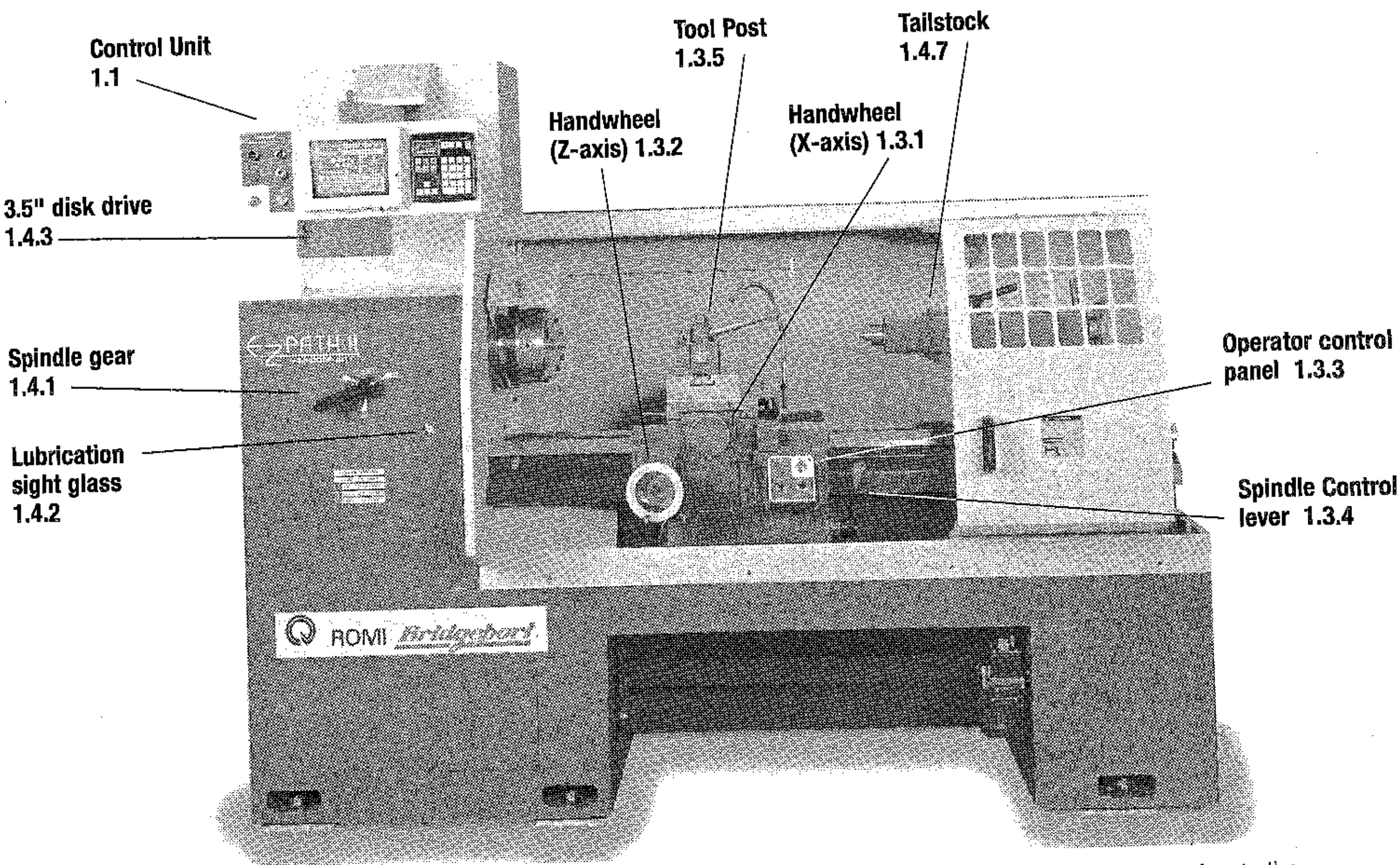


Figure 1-1 Overall view of the EZPATH II lathe. The number following the name of each control or feature refers to the numbered section of this manual which describes it.

1.1. Control Unit

The heart of the EZPATH is the control unit, mounted on a pendant on the left side of the machine. It houses the CRT (cathode-ray tube) display; a keyboard; and, to the left of the CRT display, a *control panel.* (On the tool carriage is another control panel with some of the same controls.) You can use the control unit to operate the machine and turn parts, or you can use it to create, save, and edit programs; you can rotate it to whichever position is convenient.

1.1.1. Start, Hold

Use these buttons to initiate execution of a program or Do Event command; you can use the Hold button to stop further execution of any program or Do Event command. Hold will halt all axis motion, but does not turn off the spindle. Operations that have been put on Hold can be resumed by hitting the Start button or pressing the [+ START] key on the keyboard. After hitting the Hold button, you can hit [0 EXIT] to simply terminate a program. (These keys are not on the screen when you first turn on the machine, but you'll see them when you're running a program.)

1.1.2. Jog+, Jog-

The Jog switch is used to move the tool carriage back and forth along the Z-axis. Turn the switch to the left for motion towards the spindle (the – direction); rotate it to the right to move it away from the spindle. After rotating the switch, press it in to begin moving the carriage in the selected direction. When you are in Jog mode (see section 2.2.4) you can use this switch to jog the X-axis as well.

1.1.3. Emergency Stop

The large red mushroom-shaped button is the *emergency stop* button; it is used to halt all action of the lathe in an emergency. When this button is pressed, axis motion stops, the spindle stops, and the screen displays an alarm message. See section 1.6 for emergency shutdown procedures and recovering from them.

1.1.4. Spindle Power Indicator

Only when this indicator light is turned on and glowing green is it safe to work near the spindle.

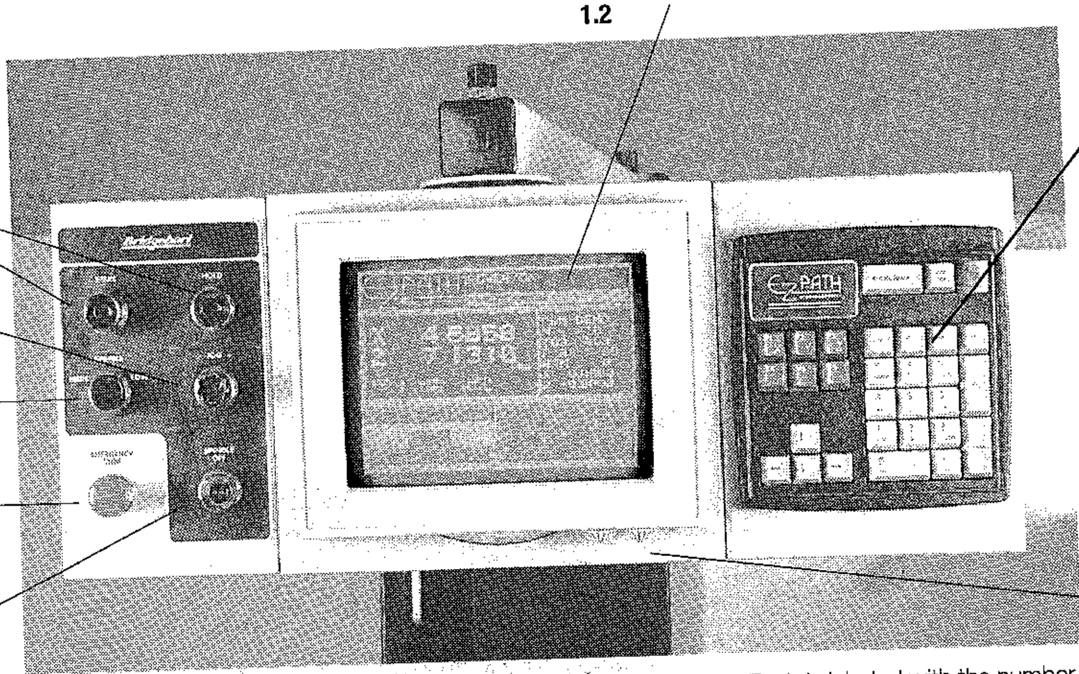


Figure 1-2 The EZPATH control unit, showing the primary operator controls. Each is labeled with the number of the section of this manual which describes it.

1.2. Computer Display

The EZPATH control unit includes a high-resolution 9" display screen. Usually, the screen is divided into two main areas, as shown in Figure 1-3. The lower part of the screen displays pictures of keys with abbreviated commands; these tell you the command assigned to each key on the keyboard. The rest of the screen displays status indicators; each gives you a particular piece of information about the current operating status of the machine, so that you can at all times know exactly what the machine is doing.

1.2.1. Tool Position

The large XZ indicators show the current position of the tool within the EZPATH coordinate system (see section 2.1 if you are not familiar with the coordinate system used by EZPATH). If the coordinates are displayed with 4 decimal places, the units are inches; if 3 decimal places are shown, the units are millimeters (section 2.2.21).

1.2.2. ABS/TMP

EZPATH gives the operator the ability to temporarily define a new coordinate system without affecting the current job set-up (see section 2.2.20). If such a coordinate system is in effect, the small indicator to the lower right of the tool position reads **TMP**; otherwise, it reads **ABS** (the default setting).

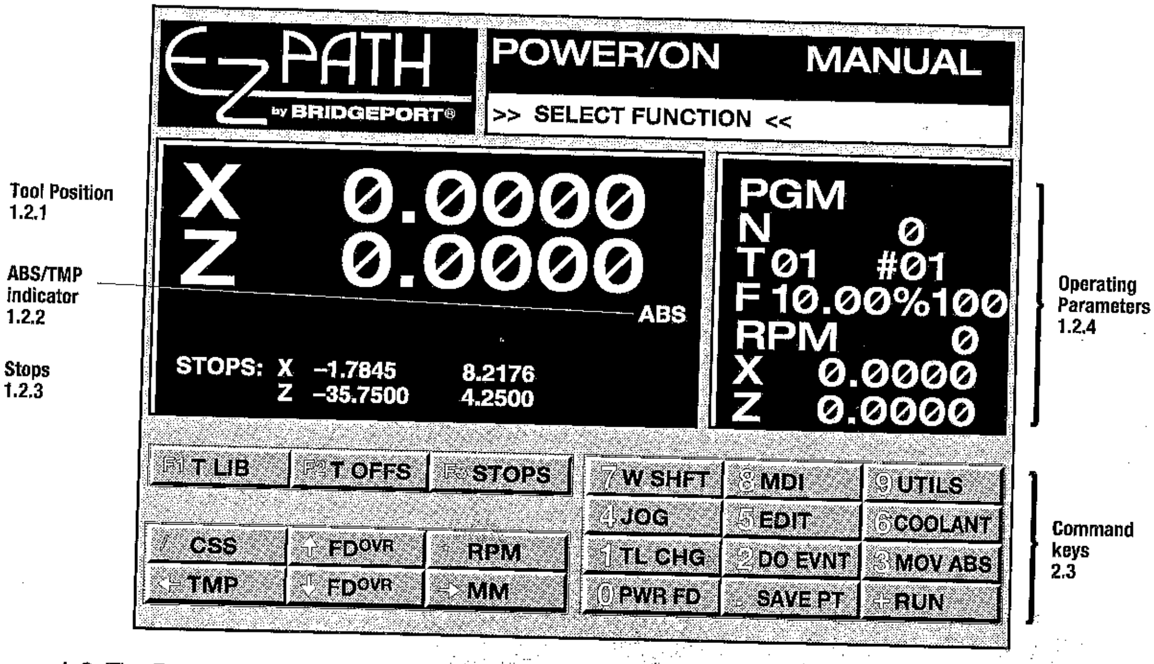


Figure 1-3 The Basic Operations screen. The keyboard pictures on the bottom half of the screen are called *softkeys*—each key represents a software command. These are discussed in Chapter 2. The rest of the screen displays different status indicators; each is discussed in the numbered section listed on the illustration.

1.2.3. Stops

These show the travel limits that you can set for movement in the X and Z axes; see section 2.2.16.

1.2.4. Operating Parameters

To the right of the main coordinate display is another box which shows a number of operating parameters.

PGM This displays the name or number of the program that is currently loaded. If no program is loaded, this area is blank.

N This is the sequence number (line number) of the current program.

T The number right after the T is the number of the currently selected tool. The number following the # is the offset number (see Chapter 3 for tool libraries and offsets). Before doing any work (unless you are operating the machine manually), you must select a tool and offset.

F This shows the feedrate in feet (or meters) per minute. It displays the actual rate of motion, not the programmed rate. It will display the rate of motion whether the tool is being moved with the handwheels or automatically; it will display RAPID when the tool is moving at the rapid-positioning rate.

Note that Do Event and MDI commands will ask that you input feedrates in inches (or mm) per spindle revolution; nevertheless, they will be converted and displayed here in feet (or meters) per minute.

The number following the % sign is the feed override percentage. Using the feed override keys, the operator can override any programmed feedrate (section 2.2.19) by a fixed percentage; if this is done, that amount is displayed here.

RPM This is the actual rpm of the spindle, not the programmed rate; the actual rpm could vary up to 10% from what is programmed. A positive number indicates counterclockwise rotation; a negative number indicates clockwise rotation.

X, Z These coordinates show the remaining distance the tool has to move to reach the destination point called for in the current instruction.

1.2.5. Screen Intensity

The EZPATH screen is subject to phosphor burn-in, which often occurs on CRT screens when they continuously display the same information. To prevent damage, use the controls mounted just below the screen to reduce the screen intensity. This is especially important if the machine will not be used for a long period of time. To help extend the life of the computer screen, a screen saver has been installed and will turn on after 3 minutes if the machine is not used. The screen saver will turn off if an axis handle is moved or a key on the keypad is struck. This first keystroke will not affect or alter any program screens or operations.

1.3. Tool Carriage Controls

1.3.1. X-axis (crossfeed) handwheel

Two handwheels are mounted on the tool carriage; the smaller one, mounted higher up, controls the crossfeed movement (X-axis); it feeds the tool into the part perpendicular to the axis of spindle rotation. Rotating it clockwise moves the tool away from the operator (in the negative direction). It is calibrated in 0.002" so that one complete clockwise revolution feeds the tool 0.5000" away from the operator (these units are diameter values).

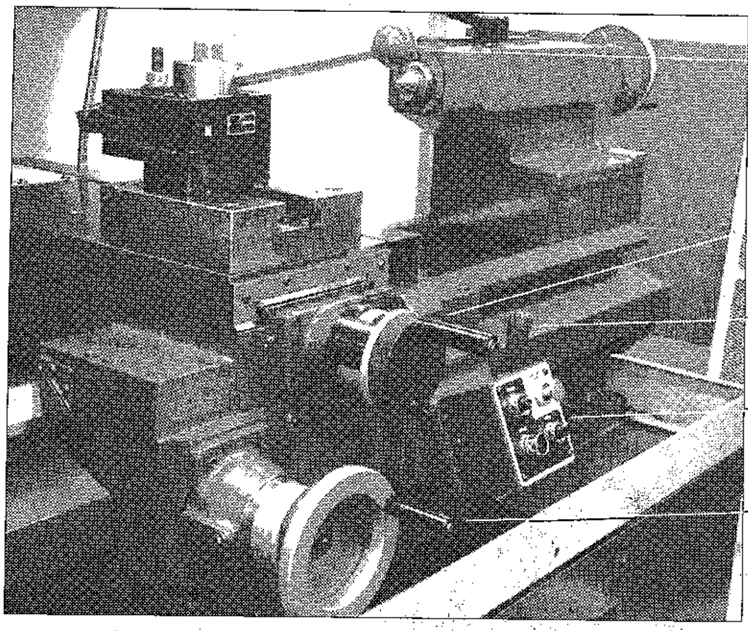


Figure 1-4 Close-up view of the EZPATH II tool carriage. The panel on the right duplicates several of the controls found on the pendant-mounted control unit, so you can easily use the machine in either position.

1.3.2. Z-axis handwheel

The larger handwheel moves the carriage to the left and right, parallel to the spindle rotation (the Z-axis). Rotating it to the right moves the carriage to the right (the positive direction). It is calibrated in units of 0.1 mm, so that one complete clockwise rotation moves the tool carriage 24mm to the right.

Both the X and Z handwheels are electronically linked to the control unit, so that any time you move the tool with the handwheels its position is automatically updated on the display.

1.3.3. Operators' control panel

To the right of the handwheels is another control panel. It has four controls, three of which are duplicates of those on the control unit: Emergency Stop, Start, and Hold. These work exactly the same way as described in section 1.1; they are duplicated here because they might be more accessible to the operator while a part is being cut.

The other control is the Select button. It is used in the execution of the Taper, Radius, and Chamfer commands in Do Event mode. In these instances, pressing the Select button slaves the X-axis to the Z-axis handwheel so that movement in both axes will occur simultaneously (see sections 5.1.4–5.1.6). Pressing the Select button a second time restores normal handwheel operation.

1.3.4. Spindle Control

On the right side of the tool carriage is the spindle control handle, 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 engages a brake which stops the spindle immediately. To reverse the spindle, move the handle to the left and down from the stop position. If you move the handle up or down from the stop position without moving it to the left, you will disengage the brake but will not start the spindle.

1.3.5. Tool Post lock

This handle locks the tool holder; pull it forward to lock the tool holder in place, and push it back to release it.

1.4. Other controls

1.4.1. Spindle Gear

On the front of the machine to the left is the spindle gear with four possible settings: neutral, and three drive gears A–C. Setting A produces spindle speeds of 4–400 rpm; B is 12–1,180 rpm; and C is from 36–3,500 rpm. Make sure the spindle is turned off (using the handle in 1.3.4) before changing gears.

1.4.2. Lubrication sight glass

The small bubble below the spindle gear is a lubrication sight glass. When the proper level of lubrication is in the lubrication system reservoir, this glass should be about half-full; if it is not, see the *Maintenance Manual* or your supervisor.

1.4.3. Diskette Drive

The EZPATH can use programs stored on 3.5" floppy diskettes, like those used on most personal computers. Make sure a diskette is not in the drive when powering up or restarting the EZPATH, or it will be unable to load the control software.

1.4.4. Power Switch

The main power switch is located on the left side of the machine. Please note that this switch provides power to both the axis-drive and spindle motors *and* to the computer control unit. The On position is indicated by an arrow; when power is turned on you will feel a solid click.

1.4.5. RS-232 port

This port (also called a *serial port*) is used to transmit data between the EZPATH and another remote computer or machine. See section 9.5 for more information on communicating with other machines.

1.4.6. Reset button

Also on the side of the machine is the Reset switch, which is used to restart the control software (called *rebooting*) without turning off the main power.

1.4.7. Tailstock

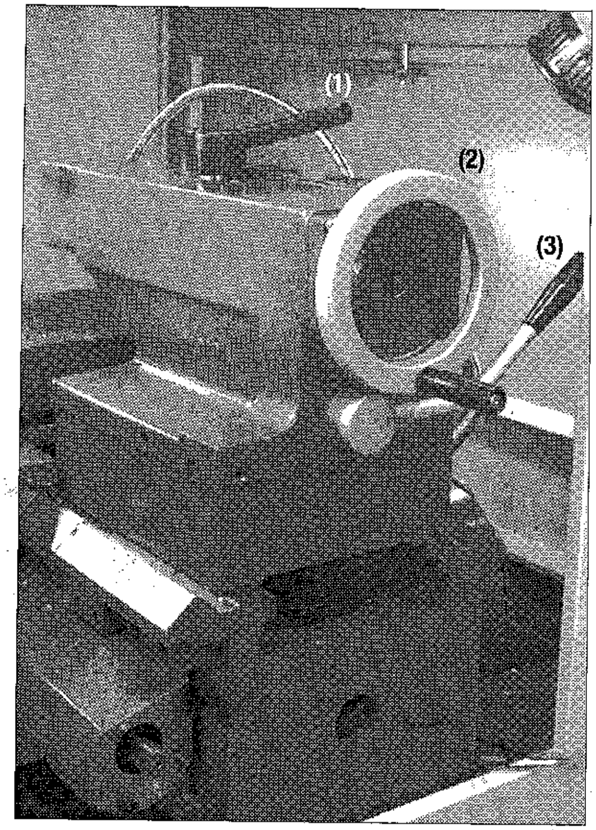


Figure 1-5 The EZPATH II tailstock. The controls are: (1) quill clamp lever; (2) quill advance handwheel; (3) tailstock clamping lever.

Figure 1-5 illustrates the EZPATH tailstock. The tailstock is entirely manually controlled. The clamping lever, located on the far side of the tailstock, clamps the tailstock in position when pulled up. The lever has a multiplier built in so that you do not need to apply very much pressure to firmly secure the tailstock. The handwheel on the rear of the tailstock advances and retracts the quill; use the lever on top of the tailstock to clamp the quill in position.

Keep in mind that, all else being equal, overall machine rigidity will be enhanced the farther to the right the tailstock is. Also, before turning on the EZPATH, make sure the tailstock is not in a position to interfere with the homing process.

1.5. Start-up Procedures

After turning on the power, you will see numerous messages flash across the EZPATH display, as the computer automatically loads the EZPATH software and performs internal diagnostic tests. In normal operation, you will not need to pay attention to these messages. The EZPATH computer will be unable to start up correctly if a diskette is in the disk drive when power is turned on.

When the EZPATH software has been loaded, you will see in the upper left of the display EZPATH NOT HOMED. HIT [MOV ABS]. This means that the axes have not been homed. Each time it is turned on—

before any work is done—the EZPATH needs to determine the limits of the carriage travel. Press the [3 MOVABS] key to select the Move command from the keypad; then press the [+ START] key to begin homing.

When the EZPATH is homing, first the tool carriage retracts toward the operator; then, it moves all the way to the right. The Basic Operations screen (Figure 1-3) is displayed when homing is complete.

It is possible to bypass the homing operation when you are restarting the machine after an emergency shutdown, and you first need to move the tool to a clear point; see next section.

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!!

1.6. Emergency Start-up

It is possible at some point that you will experience an emergency interruption of your program or machine operation. These can be of two types: an involuntary shutdown, such as a power outage, where power to the machine is unexpectedly cut off; or, a voluntary shutdown, when the operator uses the Emergency Stop button to interrupt an operation.

1.6.1. Involuntary Shutdown

To recover from an involuntary shutdown (power outage, etc.) where you need to clear the tool from the part, follow these steps:

1) Turn the main power switch to the OFF position until power is available again.

2) Move the spindle lever (section 1.3.4) to the OFF position.

3) Power up the machine again when power is available.

4) When all of the internal testing routines have been performed, you should see the message EZPATH NOT HOMED. HIT [MOV ABS] (like you usually do); at this point, do *not* home the machine. Hit the [4 JOG] key instead. The screen will read: >> HIT [MOV ABS] MOVING AXES TO HOME POSITION—the EZPATH still wants to home the axes before you do anything. 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.

5) Now you can use the jog switch on the control panel to clear the tool from the part. First, press the [↓ FD OVR] key until the feed override percentage is set to 0% (see section 1.2.4).

6) At this point, the jog switch is assigned to the X-axis; hit the the [* JOG Z] key if you need to jog the Z-axis instead.

7) Turn the jog switch to the + or – direction, as needed. While holding the switch in position, hit the [↑ FD OVR] key a couple of times; the tool will slowly begin to move in the selected direction. As the tool moves away from the part, use the [↑ FD OVR] key some more to increase the speed.

8) When the tool is clear of the part, use the [0 EXIT] key to leave Jog mode, and then hit the [3 MOVABS] key to home the machine.

1.6.2. Voluntary Shutdown

To clear the tool from a work stoppage caused by pressing the Emergency Stop, follow these steps:

1) Pull out the Emergency Stop button; hit any key to clear the alarm message.

2) If the spindle was running when you pressed the Emergency Stop, you will now see another alarm asking you to disable the spindle. Move the spindle lever (section 1.3.4) to the OFF position, and press any key to clear the alarm message.

3) Now, you can use the Jog mode to clear the tool from the part. Exit whatever operating mode you were in and return to the Basic Operations screen; hit the [4 JOG] key to enter Jog mode. Use the jog switch on the control panel to clear the tool from the part. First, press the [↓ FD OVR] key until the feed override percentage is set to 0% (see section 1.2.4).

4) At this point, the jog switch is assigned to the X-axis; hit the the [* JOG Z] key if you need to jog the Z-axis instead.

5) Turn the jog switch to the + or – direction, as needed. While holding the switch in position, hit the [↑ FD OVR] key a couple of times; the tool will slowly begin to move in the selected direction. As the tool moves away from the part, use the [↑ FD OVR] key some more to increase the speed.

6) When the tool is clear of the part, use the [0 EXIT] key to leave Jog mode.

1.7. Differences from the EZPATH I

The EZPATH II offers a number of enhancements from the EZPATH I lathe—a more powerful headstock motor, larger part capacity, etc. However, from the point of view of software and operating controls, the only real differences relate to the spindle control. Whereas on the EZPATH I the spindle speed was set via a system of gears and handles, the EZPATH II gives you much more sophisticated control capabilities via the control software. Section 1.4.1 describes the EZPATH II's simplified gear handle; sections 2.2.17–2.2.18 describe the new programmable spindle speed and constant surface speed features.

Any EZPATH I program should run unmodified on the EZPATH II, with the exception that spindle speed instructions will need to be edited to include the spindle gear; see section 6.2.6 for information on the Spindle RPM program command.

Chapter 2
Basic Operations

This chapter describes how to begin applying some of your basic lathe experience to a computer-controlled lathe like the EZPATH II. Depending on how much experience you have, you may or may not need to read all of the sections here. Section 2.1 describes how to use the EZPATH coordinate system to describe the location of the tool within the workspace. Section 2.2 introduces the EZPATH software and the commands available on the Basic Operations screen, which is what you see when you first turn on the machine.

2.1. The EZPATH coordinate system

If you've ever used a CNC machine tool—or even a lathe with a digital read-out—you've encountered *coordinate systems*. Basically, a coordinate system is a measured grid that we imagine overlaying the workspace—like a giant sheet of graph paper. There is a zero point (called the *origin*) and every point in the workspace is described in terms of its distance from the origin. If you imagine a giant sheet of graph paper laying flat over the machine, some of the lines go left–right, and some go front–back. Wherever two lines cross, there is a point. On the EZPATH, every point is identified by two numbers, called *coordinates*; they are the distance from the origin to the point in the left–right direction, and in the front–back direction. For example, a point with coordinates of (2,4) would be two units in back of the origin, and then 4 units right.

EZPATH uses the coordinate system to specify at all times the location of the tool and the location of the workpiece. The coordinate system lets you tell the EZPATH clearly, concisely, and consistently where to go when working on a part. If you are going to use the EZPATH—or any CNC machine tool—in any intelligent way, you must master working with a coordinate system.

Figure 2-1 illustrates the two major *axes* of the EZPATH coordinate system. An axis is a specialized term that loosely means direction. If you look at a graph, the two heavy perpendicular lines that cross at the origin are the axes. When the tool is being crossfed into or away from the work, we say it is moving in the *X-axis*; when the tool carriage is moving left or right, we say the tool is moving in the *Z-axis*. Each axis in any coordinate system is labeled with a letter—usually *x*, *y*, or *z*. In the case of the EZPATH the *X-axis* measures position towards or away from the operator, and the *Z-axis* measures position left and right. Again, to go to the point (2,4) means that—starting from the origin—you move in two inches (or whatever units you have chosen), and then move right four inches.

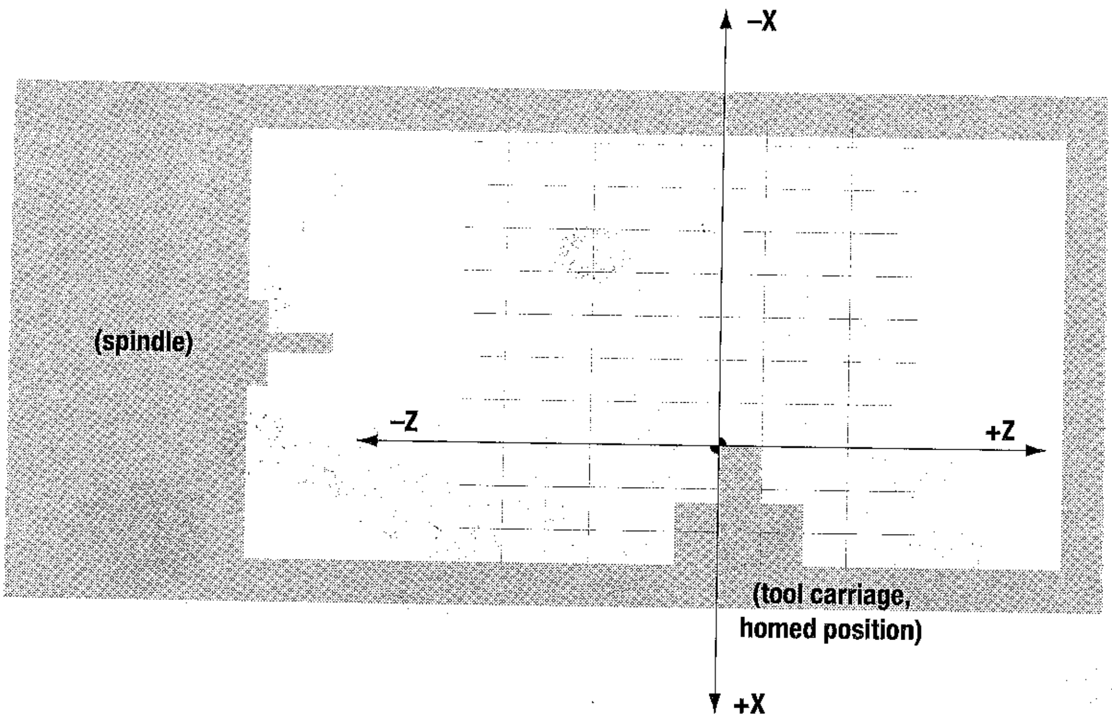


Figure 2-1 The coordinate system used by the EZPATH lathe. Notice that when the tool is moving away from the operator into the part, we say that it is moving in the –X direction; when it is moving toward the operator, the +X direction; when it is moving toward the spindle, the –Z direction; and away from the spindle, the +Z direction.

The main EZPATH display screen (see Figure 2-5) displays the X- and Z-coordinates of the tool position. Instead of using words like "up" and "down," "left" and "right," or "in" and "out," we use just two words: *positive,* and *negative.* When you crossfeed the tool into the work, you are moving it in the negative X (–X) direction; when you retract it towards yourself, you are moving it in the positive X (+X) direction. When you move the tool carriage right, you are moving it the +Z direction; when you move it left (towards the spindle) you are moving it in the –Z direction. Use the two handwheels to move the tool, and watch the coordinate display change. If you move it across the origin, you will see the coordinates begin with a "–" sign. No matter where the tool moves, EZPATH can use the coordinate system to tell you exactly where it is, using just two numbers.

Different coordinate systems may use different letters to describe the axes and directions. If you've studied geometry in high school, for instance, you're probably used to seeing the two axes as *x* and *y.* Different CNC machine tools—grinders, and mills, for example—use still other coordinate systems. The basic concepts are the same, however, and once you learn them for one system, learning other systems will be easy.

2.1.1. Working with coordinates

Figure 2-2 illustrates a coordinate system like the one used on the EZPATH. The two axes and their directions are labeled; the units of measurement are represented by the grid lines. The coordinate system has been superimposed on the image of a lathe to help you relate the coordinates and directions to an actual machine. The origin (point 0,0) is represented by the ⊕ symbol.

Try this exercise now. Take a pencil and move from the origin to each numbered point in order; write down the coordinates of each point, and count the number of units you had to move in each direction to get to the point from the previous point. Check your answers at the end of this chapter. Don't forget to use "–" if you are moving in the negative direction.

Notice that on coordinate system diagrams like this, –X appears to be up, and +X appears to be down. This may take some getting used to. Remember, though, that the X direction isn't really up and down, but towards and away from the operator; when you see a grid representing the EZPATH coordinates, try to remember that you are looking at it from the top of the machine, not the side.

To use Point 1 for an example: to get to point 1, you have to move 3 units in the –X direction, and then 4 units in the +Z direction. Its coordinates, therefore, are (–3, 4); since we were at the origin when we began moving, the "Distance from Previous Point" is the same as the coordinate position.

	Position	Distance from Previous Point	
Point	**(X , Z)**	**(X , Z)**	
1	(-3 , 4)	(-3 , 4)	[from origin]
2	(___ , ___)	(___ , ___)	
3	(___ , ___)	(___ , ___)	
4	(___ , ___)	(___ , ___)	
5	(___ , ___)	(___ , ___)	

2.1.1.1. Absolute vs. incremental

Many EZPATH operations let you specify points as either absolute or incremental coordinates. Absolute coordinates are what we've been talking about all along; it means that each coordinate represents a distance from the origin—like the "Position" coordinates you wrote out in the exercise in 2.1.1.

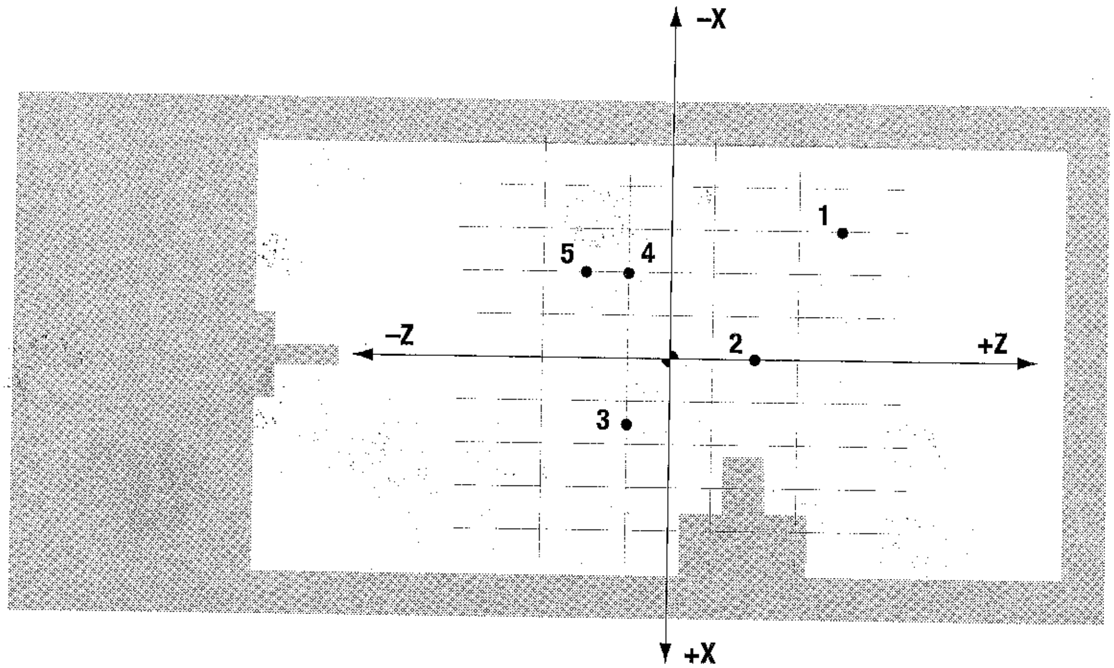


Figure 2-2 Find the coordinates of each point to familiarize yourself with using the EZPATH coordinate system.

Incremental coordinates, by contrast, are measured from the current point. In the exercise in 2.1.1, the "Distance from" coordinates you figured out are incremental coordinate positions. The absolute coordinates of point 2 are (0, 2); but the coordinates of (3, –2) are incremental coordinates relative to point 1. Using incremental coordinates, you can easily program a dimensioned shape or contour and then worry later about where the origin will be.

In general, whenever you see a set of coordinates, you can assume that they are absolute unless it is clearly stated otherwise.

2.1.2. Layers of Coordinate Systems

When you are operating the EZPATH, you will actually use several coordinate systems, layered one on top of the other. The set of axes will always remain the same—what makes each system different is that each one has a different origin.

As we mentioned before, every time you see a set of coordinates, they represent a distance from an origin point. But who decides where the origin point is?

When you first power up the machine, EZPATH asks you to "home the axes" (section 1.5). This means that it pulls the tool carriage out as far as it can go, and then moves it all the way to the right. This tool location, by definition, is the origin (point 0,0) of the *Machine*

Coordinate System. This is the coordinate system that was illustrated in Figure 2-1, and is the lowest-level coordinate system.

You can use the Work Shift command (section 2.2.8) to move this origin to a new point. Whatever origin you set with Work Shift is the origin of the *User Coordinate System*. Most of the time, you will not use this feature at all—in those cases, the User Coordinate System will be the same as the Machine Coordinate System.

Once you've homed the axes, EZPATH knows where the tool is based on the travel limits of the machine. Now the software needs to know where your part is in relation to the tool. This is what setting *offsets* is all about (section 3.2). Basically, before beginning work, you move the tool so that it touches the face or diameter of the stock. EZPATH keeps track of how far it travels to do this, and records the distance as a tool offset. EZPATH uses the offset to set up a new origin at this point—the beginning of your part. This new origin defines the third coordinate system—the *Part Coordinate System*, illustrated in Figure 2-3. This is the coordinate system you will use for most of your regular work. Since the origin is the beginnning of the part, it is easy to transfer information from a blueprint to a part program.

Let's review for a minute. We began with the *Machine* Coordinate System; the origin of this system is the homed position of the tool. If you do not set a Work Shift, this coordinate sys-

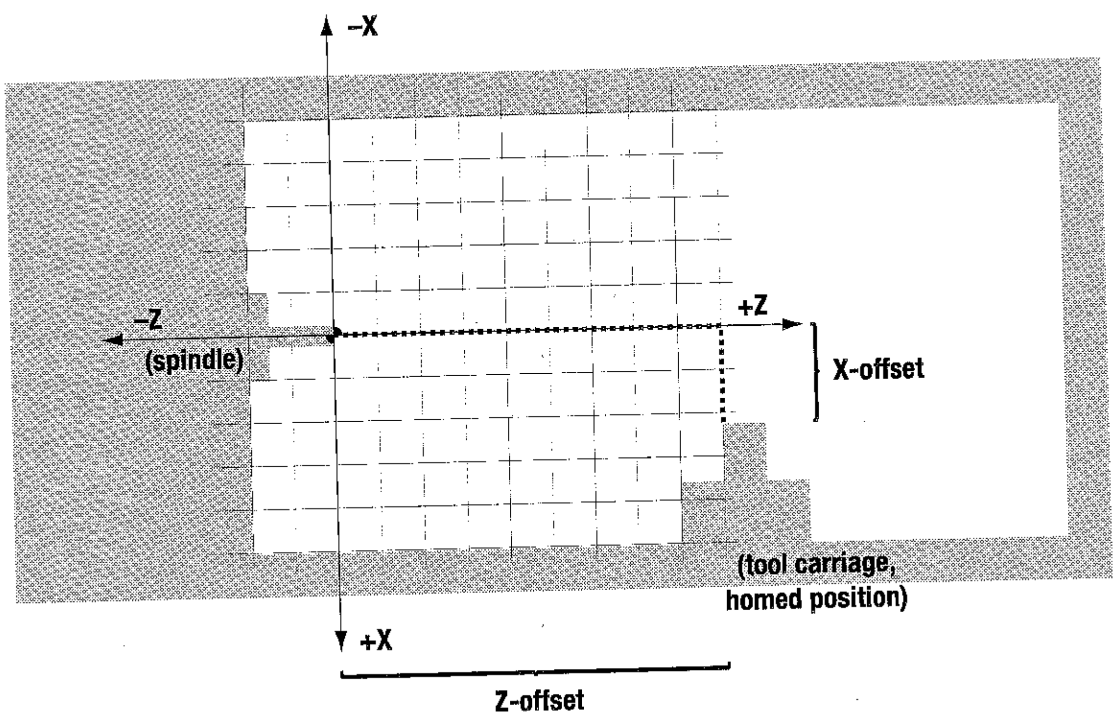


Figure 2-3 Notice how offsets are used to move the origin of the coordinate system to the beginning of a part.

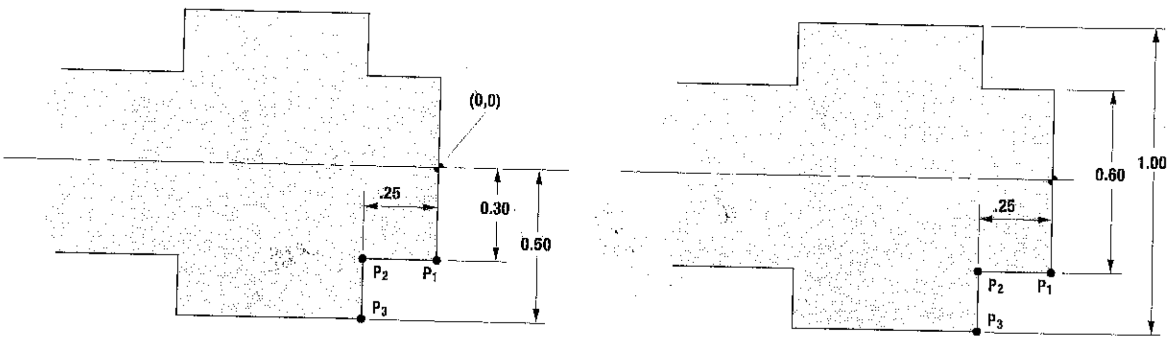


Figure 2-4 X-axis positions on the EZPATH are dimensioned in diameter values, like the drawing on the right. In the left drawing, without using diameter values, the coordinates of the three points would be: $P_1 = (.3,0)$; $P_2 = (.3, -.25)$; and $P_3 = (.5, -.25)$. The same three points in diameter values would be: $P_1 = (.6,0)$; $P_2 = (.6, -.25)$; and $P_3 = (1, -.25)$.

tem is identical to the *User* Coordinate System. You then adapt this system to a specific part (and a specific tool) by setting offsets. When you activate an offset, the origin changes by the amount of the offset and you have created a *Part* Coordinate System.

The large coordinate display on the Basic Operations screen displays the coordinates of the current tool position in the Part Coordinate System. To see the coordinates of the same point in the User Coordinate System, select the Tool Offsets command (F2 T OFFS key); in the lower right corner of the screen you will see **UCS LOCN** followed by the User Coordinates of the current point. If the active offset has a value of (0,0), then the two sets of numbers will be the same. Note that, to go from the User Coordinates to the Part Coordinates, you need to *subtract* the amount of the offset.

Note that once you make an offset or work shift active, it remains in effect until you select a new one, even if you restart the machine. That is why, after homing the axes, the coordinates on the EZPATH display screen might not read (0,0)—an offset or work shift that was set earlier is still active.

2.1.2.1. Diameter values

Generally speaking, when you are turning a part, the origin of the X-axis will be the centerline of your part. However, if you're used to working with a lathe, you're used to seeing diameter values on your blueprints: in other words, parts aren't dimensioned from the center outwards, but across the entire diameter. How can you reconcile these two different ways of looking at things? The answer is to *scale* the X-axis so that it is dimensioned in diameter values: if you start at the origin and move the tool outwards by 1 inch, you will see the coordinate display increase by 2 inches! Figure 2-4 illustrates the difference. Normally, you do not

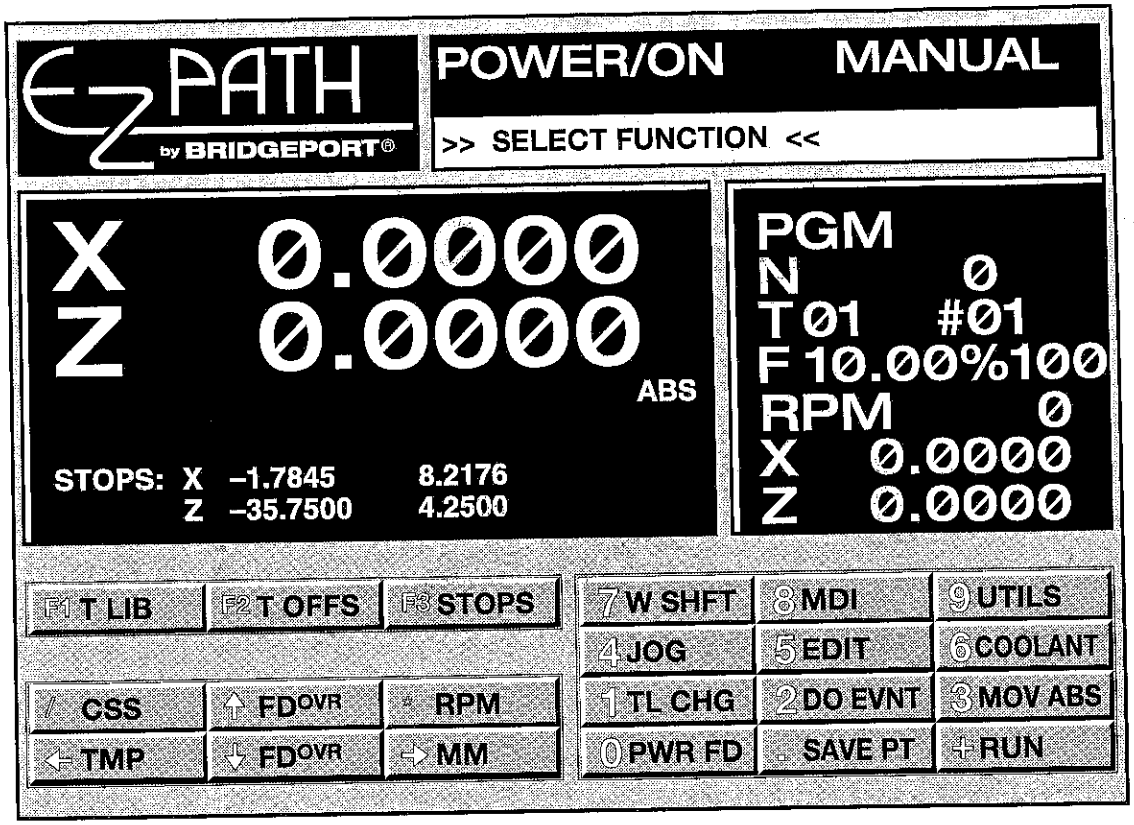


Figure 2-5 The Basic Operations screen is the "home base" of the EZPATH software. Here you can perform some basic machine set-up and operation tasks, and also switch to other operating modes.

need to think about this too much; since everything is consistent—even the gradations on the handwheel—you usually won't notice. Whenever EZPATH is asking you to input information or coordinates, it will ask for **DIA** when it is expecting diameter values, and **X** when it expects a non-diameter value.

2.2. Using the EZPATH software

Figure 2-5 illustrates the Basic Operations screen—the screen that you first see after powering up and homing the machine. The Basic Operations screen is the "home base" of the EZPATH software; from here, you can access all of the other operating modes. On the bottom of the screen, you can see pictures of computer keys, each of which is labeled with a number and a command abbreviation. These pictures correspond to the keyboard on the control unit, letting you know at all times what EZPATH command is assigned to each key. EZPATH uses *softkeys*—meaning that the function assigned to each key changes depending on where you are in the program. By using the pictures on the screen, however, you will always be able to tell which key does what.

All of the commands that produce machine operations operate in a consistent way. First, you select the command from the keyboard; then, you enter any required data; then, before the command is actually executed on the machine, you have the opportunity to position the tool, turn on the spindle, adjust the feedrate, etc. The command will not be executed until you hit the [+START] key.

While you are entering data, you can use the BACKSPACE key and the arrow keys to edit entries; hit [ENTER] or the ↓ key to move from one field to the next. Hit [ENTER] or the ↓ key while you are in the last field to complete the data entry.

EZPATH provides a number of aids to help you calculate necessary program data. A standard calculator is built in, and also a special calculator designed to help solve geometry problems; these are described in detail in Chapter 10. Where these are available, you will see appropriate buttons ([/GEOM], [*CALC]) displayed directly in the dialog box. You might also see a [+INC] button; this changes the mode of the entered coordinates from absolute to incremental. A [+TLIB] button will display pictures of all the tools in the tool library. (A *dialog box* is a box that pops up on the screen and asks you to type in some information. Figure 2-6 is a dialog box.)

At any time while a dialog box is displayed on the screen, you can cancel the entire operation by pressing the [ESC] key.

One final note: as you move through the software, you will notice that some commands are repeated in different places. For example, Work Shift is displayed on key 7 in Basic Operations; but if you move to Jog mode, you will also see it displayed on key 1. This is done so that wherever you are in the program, you will have a complete set of appropriate commands available, without having to constantly jump back and forth. The remainder of this chapter describes the Basic Operation software commands.

2.2.1. Tool Change

[1 TL CHG]

The Tool Change command lets you select a new tool from the tool library and/or activate a new offset; the [+TLIB] key shows you all of the tools in the tool library, labeled by type and tool ID number.

When you select the Tool Change command, you will be asked to specify the **TOOL NO** (tool number) and the **OFFSET NO**. The "tool number" is a number that you assign to the tool based on a particular job's requirements; for example, the sequence in which tools will be used in cutting a part. In many cases, you will select a tool number to match the offset number, so that, for example, tool number 3 uses offset number 3 (see chap-

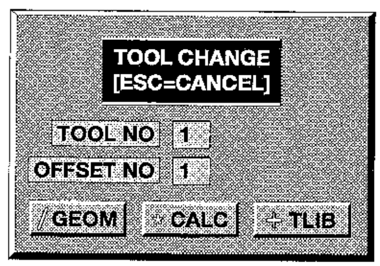


Figure 2-6 Use the Tool Change command to to either change the tool or the active offset. To activate any offset, you must use this command.

ter 3 for more information on Tool IDs and offsets). Note that the tool number is *not* the same as the tool ID; if you hit the [+TLIB] key to look at the tools, it will also show you the tool ID number, but you will not use the tool ID here.

This is also the command to use when you just want to activate a particular offset, and not change the tool at all; simply hit [ENTER] to get through the **TOOL NO** box, and then enter an offset number. When you select or change an offset, it is *your responsibility* to ensure that it is the proper one. Selecting an incorrect offset can cause the tool to crash the machine or part; this could cause serious injury to you, and damage to the machine.

WARNING!

2.2.2. Do Event

[2 DO EVNT]

This places you in Do Event mode, which allows you to execute computer-controlled operations one at a time. See Chapter 5 for a full description.

2.2.3. Move (Absolute)

[3 MOVABS]

Use this command to make a rapid (non-cutting) move to a specified point. When you select the command, it will ask you to specify **DIA ABS** and **Z ABS** : in other words, absolute X and Z coordinates, relative to the (0,0) position on the screen. By pressing the [+INC] key, you can move incrementally, to a coordinate position which is offset from the current tool location. This command produces *vector* motion; i.e., the tool moves on a straight line directly to the point, not axis-by-axis.

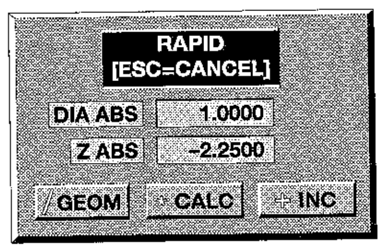


Figure 2-7 The Move command produces rapid motion to a specific point—in either incremental coordinates, or absolute.

2.2.4. Jog

[4 JOG]

Selecting this command places you in Jog mode. Figure 2-8 displays the Jog mode screen. Think of Jog mode as a "command center" for positioning the tool with the greatest possible flexibility and accuracy.

Jog mode gives you the following capabilities:

1) While you are in Jog mode, you can use the jog switch on the control panel (section 1.1.2) to move along either axis. Recall that when you are *not* in Jog mode, the switch is

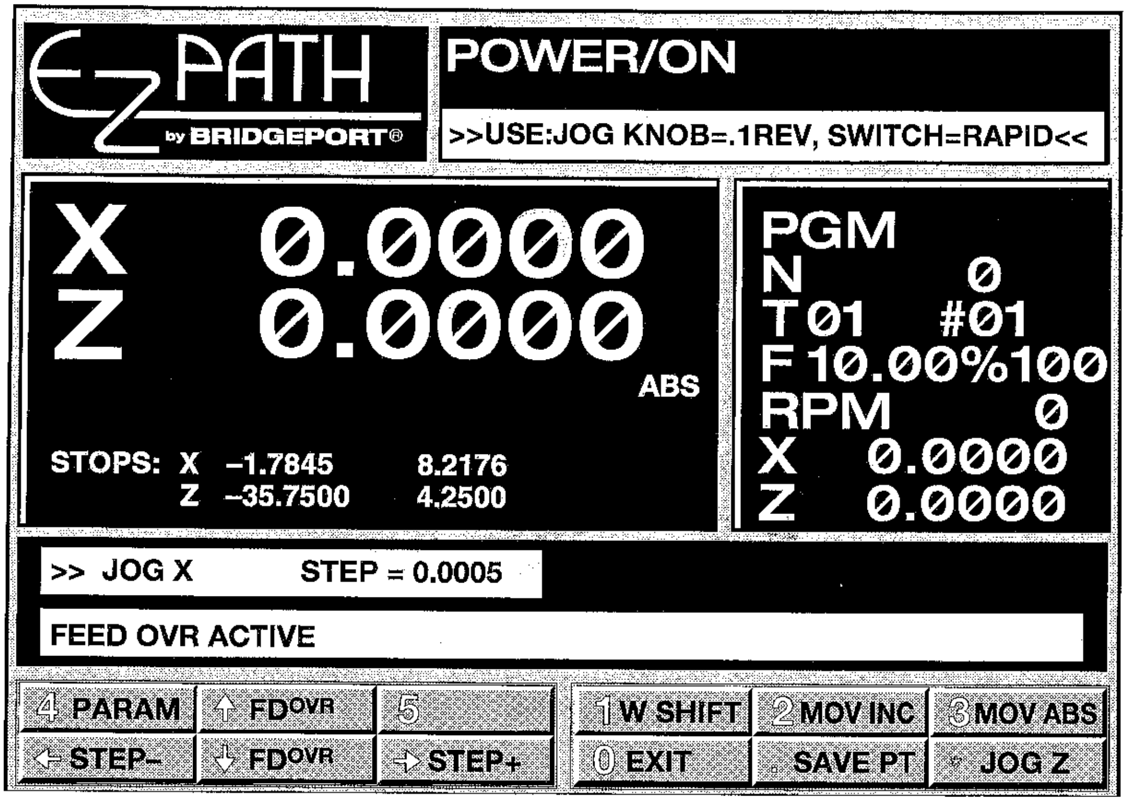


Figure 2-8 Jog mode brings together in one place many different commands related to moving and positioning the tool.

assigned to the Z-axis. When you enter Jog mode, though, it is assigned to the X-axis. You can use the [• JOG Z] key to toggle back and forth between the X- and Z-axis.

2) You can use the [← STEP-] and [← STEP-] keys to move either axis in single steps of .0005". If you are in metric mode (section 2.2.21), the Jog step will *still* be .0005"—the display will just show the equivalent distance in millimeters.

3) You can use the Feed Override keys ([↑ FD OVR] and [↓ FD OVR]) to adjust the effective travel speed as a percentage of the programmed jogging rate (which is set with the Parameters command—section 2.2.5). Hitting the [↑ FD OVR] key will increase the speed in 5% increments; hitting the [↓ FD OVR] will decrease it in 5% increments. The Feed Override keys will work on either the jog switch or the STEP keys.

While in Jog mode, of course, you can also use the axis handwheels if you wish.

Note that the jog *keys* ([← STEP-], and [← STEP-]) are different from the jog *switch* on the control panel (section 1.1.2). The jog keys work only in Jog mode; the jog switch is always active. The jog keys move the tool .0005" each time they are pressed; the jog switch will cause continual motion for as long as the switch is turned. However, the Feed Override keys affect each identically.

While in Jog mode, you can also execute other, related commands as described below:

Work Shift command [1 W SHFT]

Use this command to define a new coordinate system origin at a specific point. It functions identically to the Work Shift command described in section 2.2.8.

Move commands [2 MOV INC] [3 MOV ABS]

Use these commands to move to a specific point: to either an incremental coordinate position relative to the current tool location (MOV INC) or to an absolute coordinate address, relative to the origin (MOV ABS). The tool will move at the rapid traverse rate—not the jogging rate—as adjusted by the Feed Override keys. These are identical to the regular Move command (section 2.2.3).

Set Parameters command [4 PARAM]

Use this command to set various system parameters—including the jogging speed. It is described in the next section (2.2.5).

To exit Jog mode and return to the Basic Operations screen, select [0 EXIT].

2.2.5. System Parameters

[4 JOG], [4 PARAM]

Use this command to establish several important system settings while you are in Jog mode. To display the dialog box shown in Figure 2-9, enter Jog mode from the Basic Operations screen and press the [4 PARAM] key. You can set values for the following parameters:

SPINDLE MAX RPM Enter here a "clamp" value for the spindle speed. While there are other places within the software where you can define a maximum spindle speed (in particular, wherever you can set constant surface speed), the value set here will override any value set anywhere else.

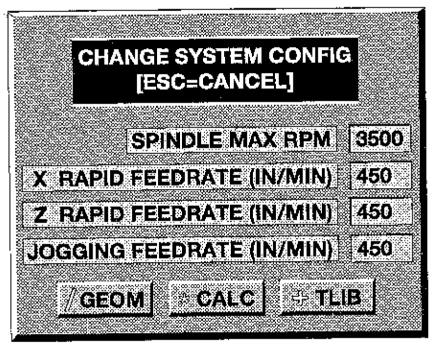


Figure 2-9 Use the System Parameters to customize your machine's performance settings.

X RAPID FEEDRATE (IN/MIN) Enter here the rate that will be used for rapid positioning moves in the X-axis, up to a maximum of 450 ipm.

Z RAPID FEEDRATE (IN/MIN) Enter here the rate that will be used for rapid positioning moves in the Z-axis, up to a maximum of 450 ipm.

JOGGING FEEDRATE (IN/MIN) Enter here the rate that will be used when the Jog keys or Jog button are used, again, up to a maximum of 450 ipm.

Note how the Feed Override keys will affect each of these speeds. With the rapid traverse rates, the Feed Override keys can only be used to adjust the rate downward. With the jogging rate, the Feed Override keys can be used to adjust it up or down, but they will adjust it upward only until the rapid traverse rate is achieved. The rapid traverse rates thus act as a ceiling on any tool-motion speeds.

2.2.6. Edit Programs

5 EDIT

Use this command to enter Edit mode, which allows you to edit part programs that have been previously created on the EZPATH. See Chapter 7 for a full description of the EZPATH Editor.

2.2.7. Coolant

6 COOLANT

After selecting this command, enter 1 to turn on the flood coolant, or 0 to turn it off. The coolant system is interlocked with the spindle, so that the coolant flow will not actually begin unless the spindle is turned on.

2.2.8. Work Shift

7 W SHFT

Use the Work Shift command to set a new origin for the User Coordinate System. When you select the command, it will prompt you for a set of coordinates; the machine will set these as the current User Coordinate position. Typically, you would enter "0" for one or both axes to define a new origin. Remember, though, that the coordinate position on the main DRO will be the coordinates you just typed in minus the value of whatever offset is active. (Work Shift affects the *User* Coordinate system; the main DRO displays the *Part* Coordinate system—see section 2.1.2 if you are unclear on the difference).

When you use the Work Shift command to change the User Coordinate origin, you are basically picking up all of your coordinate settings—offsets, stops, etc.—and moving them as one piece. You can use Work Shift, for example, to adjust an entire job set-up to a different piece of stock so you can easily cut multiple parts.

The work shift you set will be in effect until it is changed with another Work Shift command, even if you restart the machine.

Note that there is no way to recover a work shift in order to cancel it. To cancel a work shift, move the axes to the home position; select the Work Shift command; and enter (0, 0).

2.2.9. Create programs with MDI

[8 MDI]

This command lets you enter MDI (Manual Data Input) mode, which lets you create part programs using the EZPATH's easy-to-use conversational interface. With MDI, you don't need to remember any complicated programming commands or codes—you can build an entire program just by selecting commands from the screen. See Chapter 6 for details.

2.2.10. Utilities

[9 UTILS]

The Utilities command gives you access to a number of file management functions, including sending and receiving files to/from another computer. They are fully described in Chapter 9.

2.2.11. Power Feed

[0 PWR FD]

Use the Power Feed command to move the tool in a selected direction along one axis only. It will continue in this direction until it reaches either a stop or a machine limit. The directions (illustrated in Figure 2-10) are 2 = +X; 8 = -X; 4 = -Z; and 6 = +Z. After selecting a direction, enter a feed rate in inches per spindle revolution. To stop the tool yourself, press the Hold button, or use the the [ESC] key on the keypad.

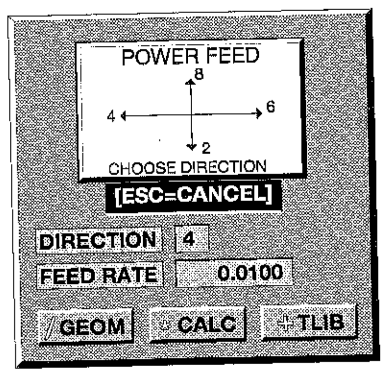


Figure 2-10 Before using Power Feed, the spindle must be turned on.

2.2.12. Save Point (Teach)

[. SAVE PT]

This is an alternate means of programming the EZPATH; it allows you to select and save up to 100 points which can be replayed at a later time. See section 6.4 for a full description.

2.2.13. Run a Program

+ RUN

Selecting this command places you in Run mode, which is used to run or preview a program. See Chapter 8 for a full description.

2.2.14. Tool Library

F1 TLIB

Use this command to maintain the Tool Library, a library of information about different tools. This information is needed by part programs so that programmed shapes can be cut accurately and precisely. See section 3.1 for a complete description.

2.2.15. Tool Offsets

F2 T OFFS

As mentioned in section 2.1.2, EZPATH uses tool offsets to locate a part within its coordinate system; use this command to access the Tool Offsets window and manage your offsets data. See section 3.2 for a complete description.

2.2.16. Stops

F3 STOPS

Use this command to set stops in either direction along either axis. Stops are disabled when a program is running or when the machine is homing; if you home the machine and the axes travel beyond an active stop, you will get an error message.

The Z-axis stops are labeled as *left* and *right.* The left stop prevents motion in the -Z direction; the right stop prevents motion in the +Z direction. The X-axis stops are *far* and *near* diameter. The far stop prevents motion into the part (-X direction); the near stop prevents motion away from the part (+X direction). For example, you can use a far stop to prevent cutting too far into the part during an OD operation, and a near stop for an ID operation. Setting stops takes two steps: first, enter the stop coordinate; then, in the **ACTIVATE** box, enter a "1" to turn it on.

Regardless of what stop values you set, the EZPATH has absolute hard-wired machine travel limits. If you try to enter a stop value outside these limits, your stops will be *clipped;* that is, the coordinate will be automatically changed to match the travel limit.

Using stops with different offsets The stop coordinates always take into account the active offset, so that if you activate a different offset or tool, the physical location of the stop doesn't change. You can see the stop coordinates—as displayed on the main DRO—change whenever you select a new offset.

Using stops with Work Shift As we mentioned in the Work Shift section (section 2.2.8), the Work Shift command is used to pick up and move an entire job set-up, including stops. Therefore, if you change your coordinate position with Work Shift, the physical location of the stops *will* change, but the coordinates (as displayed on the DRO) will not.

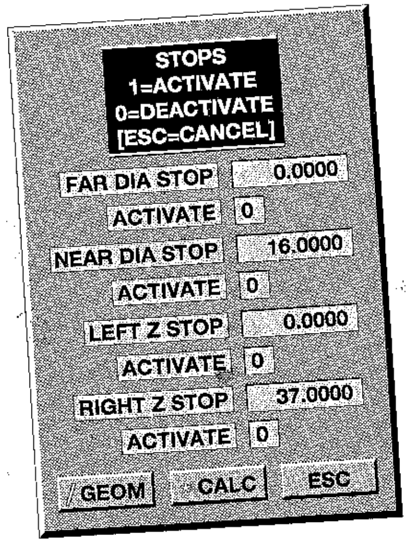


Figure 2-12 After inputting a value for a Stop, remember to enter a 1 in the **ACTIVATE** box to turn it on.

The Work Shift command can produce two types of errors with respect to stops. If the Work Shift moves the coordinates of a stop outside the physical travel limits, the stop would be clipped as described above. The second type of error occurs when the coordinate you enter in the Work Shift dialog box is outside an existing stop; in this case, the EZPATH de-activates the stops, and you need to reset them.

When the EZPATH is being homed, it will automatically deactivate any stops that would prevent it from moving to the homed position; you must re-activate them yourself.

All stops are automatically disabled when you run a program.

2.2.17. Constant Surface Speed

/ CSS

Use this command to set the EZPATH in constant surface speed mode. In this mode, the spindle speed automatically adjusts as the diameter of the workpiece changes, so that the speed of the workpiece moving past the tool tip is unchanged. Surface speed is measured in surface-feet per minute (*sfpm*); in other words, 100 sfpm means that the surface of the workpiece is moving past the tool tip at a rate of 100 feet per minute. You are prompted to enter 3 pieces of information:

Figure 2-13 Note that the EZPATH has no electronic knowledge about what the spindle gear is; you must tell it here if it is to operate properly and safely.

GEAR The EZPATH spindle has three gears. Use this setting to tell the EZPATH what gear has been selected with the handle on the front of the machine (see section 1.4.1). Gear A has a range of 4–400 rpm; Gear B covers 12–1,180 rpm; and Gear C has a range of 36–3,500 rpm. Enter 1, 2 or 3 for A, B, or C, respectively.

Note: entering a value here does *not* change the spindle gear. To place the spindle in the proper gear, you must change the gear manually with the handle on the front of the machine. All you are doing here is telling the EZPATH control what gear is in effect, so that it can issue the proper spindle-speed instructions to the machine.

CSS VALUE Enter here the surface speed you want EZPATH to maintain, in surface-feet per minute (if you are in metric mode, use meters/minute).

MAX RPM It is important to enter a maximum spindle rpm here; as the diameter of your part decreases, the spindle speed will increase, theoretically approaching infinity as your part gets smaller. This could create dangerous operating conditions leading to serious operator injury! Enter a value here that will preserve safe operating conditions; take care also to account for the rating of the chuck you are using.

It is important to understand that EZPATH figures your part diameter—and hence, the spindle speed it needs—based on the X = 0 position. This position is determined entirely by the active offsets and work shifts; EZPATH does not use a "hard-wired" spindle centerline. Therefore, to use the Constant Surface Speed feature, you must have your offsets and/or work shifts set so that X = 0 corresponds to the centerline of the spindle. If you move the X = 0 position after setting CSS—either by changing the offset, a work shift, or switching to TMP mode—you will corrupt the EZPATH's spindle-speed calculations.

To turn off constant surface speed, use the Set RPM command (next section) to set the spindle to a fixed speed.

2.2.18. Set RPM

[* RPM]

Use this command to set a fixed spindle speed. Make sure that the speed you set here is consistent with the gear that has been selected with the spindle-gear lever on the front panel (section 1.4.1). You can use a special surface speed calculator (press the [+SFPM] key) to calculate the spindle rpm that will produce a desired surface speed for a particular diameter. Note that this is *not* the same as setting a constant surface speed, as described in the previous section.

Setting a spindle speed here will turn off constant surface speed, if that is the current operating mode.

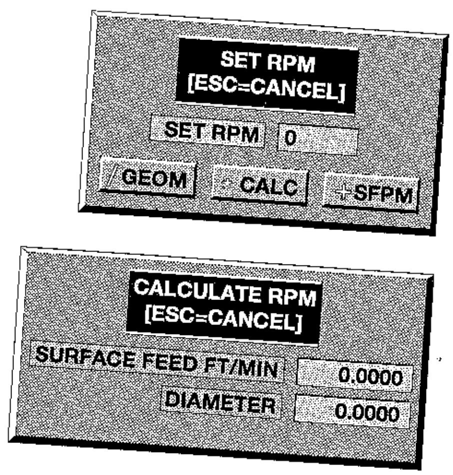


Figure 2-14 After selecting the RPM command, you can use the special surface-speed calculator (lower box) to calculate a proper speed for your particular application. The calculated value will be inserted automatically into the RPM command box.

2.2.19. Feed Override keys

[↑ FD OVR], [↓ FD OVR]

Use these keys to set the actual feedrate as a percentage of whatever is programmed. You can specify that the actual feedrate be anywhere from 0–150% of the programmed value. Each key increases or decreases the feedrate by 5%; you can see the effect in the status window (see section 1.2.4). Any modification you set here will be in effect until it is changed, even if you do things like run a program.

Note that the Feed Override percentage affects not only "feedrate" (i.e. cutting) moves, but also jogging and rapid positioning moves. Section 2.2.5 describes how the Feed Override keys work with rapid traverse and jogging rates.

2.2.20. Use Temporary coordinate system

[← TMP], [← ABS]

Use this command to break into the current coordinate system set-up and define a new, temporary coordinate system. This new coordinate system overrides all active offsets and work shifts.

When you select the [← TMP] key, the indicator to the lower right of the main coordinate readout switches from **ABS** to **TMP**. The 7 key will now display the **SET XZ** function; if you select this command, you will be asked to specify X- and Z-coordinate values that will immediately become the coordinates of the current tool position. Typically, you will use this function to enter a zero on one or both axes and establish a temporary origin. All tool moves and stops will now be displayed relative to this new origin. When you press the ← key again ([← ABS]), you will return to your regular coordinate system with its offsets. If [7 SET XZ] is not used to set a new origin, the TMP mode will be no different than the ABS mode. If you return to TMP mode, EZPATH will restore the last TMP origin.

Note the difference between TMP and Work Shift: Work Shift sets a new origin for the *User* Coordinate System, while TMP sets a new origin in the *Part* Coordinate System. Thus, you can still use all of your tool offsets after using Work Shift, but TMP simply overrides them. (See section 2.2.8 for Work Shift).

Be careful about switching to TMP while Constant Surface Speed mode is active; setting a new X = 0 position will corrupt EZPATH's spindle-speed calculations.

2.2.21. Set inches or mm

[→ MM], [→ IN]

You have the option of using either inches or metric units. If the coordinates are displayed with 4 decimal places, then EZPATH is using inches; if 3 decimal places are shown, EZPATH is using millimeters. When writing a program, you must select the proper mode here, *before* creating your program. The label on the key displays what mode it will change into, so that when the key reads [→ MM], pressing it will place you in metric mode; when it reads [→ IN], pressing it will place you in inch mode.

2.3. Answers to exercises

2) (0,+2) (+3,-2)

3) (+1.5,-1) (+1.5,-3)

4) (-2,-1) (-3.5,0)

5) (-2,-2) (0,-1)

Chapter 3
Using Tools

In the operation of your EZPATH II lathe you will need to use any number of different tools. Properly organizing and managing the information associated with these different tools is a vital part of using the EZPATH properly, effectively, and safely.

Information about tools is stored in two main places: the Tool Library, and the Tool Offsets file. Both of these are computer files that the EZPATH maintains on its hard disk. The Tool Library stores a number of pieces of information about the size and shape of each tool, including what type of tool it is—drill, grooving tool, etc. The Tool Offsets file is a table of information describing the distances from particular tools to your part; it is a way of relating the tool and part in the coordinate system so that your programs can function properly. The Tool Library is described in section 3.1; tool offsets are described in section 3.2.

3.1. Tool Library

F1 TLIB

The Tool Library is used to specify the tool type; touch-off point; and information about the size and shape of the tool. Each tool is also assigned a Tool ID when it is entered into the Library, so EZPATH can look up the information for each tool when your program calls for it.

Note that the *tool number* and the *tool ID* are two different things. The tool number is for your use only, as an operator; the EZPATH software does not associate this number with any information in the Library. Assign tool numbers to help organize the planning and set-up of a job: for example, the first tool to be used might be assigned tool number 1, the second tool number 2, etc. Typically, you will associate the tool number with an offset number so that, for instance, tool number 3 uses offset number 3. If your tools are mounted in a turret, the tool number might also equal the turret position. The Tool Change command in Basic Operations (section 2.2.1) will ask you to specify the tool number to select a tool.

When you select a tool in MDI, though, (section 6.2.7), you have to specify *both* a tool number and a tool ID. In this case, the tool number is only used to prompt the operator to install the proper tool when the program calls for it; the tool ID is used to look up the right tool information in the tool library so EZPATH can calculate the proper tool path.

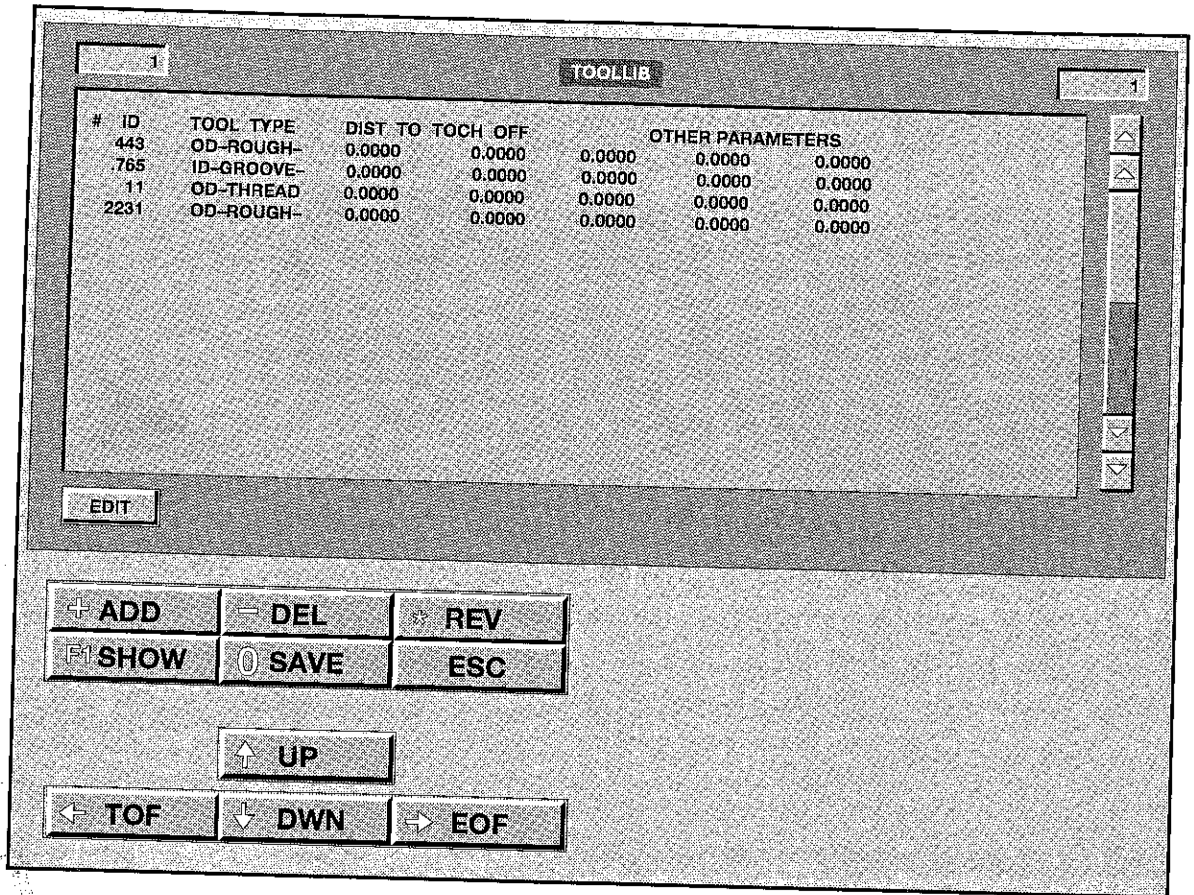


Figure 3-1 The Tool Library stores the information about each tool that is needed by EZPATH programs.

The Tool Library stores tool information in a file on the EZPATH hard disk. The Tool Library gives you a number of functions to add or delete tools from this file; modify the settings for a tool; or view a picture of the tool which is based on the Tool Library settings.

To access the Tool Library, hit the [F1 TLIB] key from the Basic Operations screen. You will see the screen shown in Figure 3-1. The main window lists all of the tools in the file. Use the cursor (arrow) keys to move up or down through this list to select a tool to view or modify: use [↑ UP] or [↓ DWN] to move up or down a line at a time, or [← TOF] and [→ EOF] to jump to the beginning or end of the list. As you move up and down through the file, the small window in the upper left corner tells you what line number you are on. The Tool Library's other commands are described below.

3.1.1. Add a new tool

[+ ADD]

Before using this command, notice where the cursor is placed in the main window; this is the line where the information for the new tool will be inserted. Use the ↑ and ↓ keys to move to a different line, if desired.

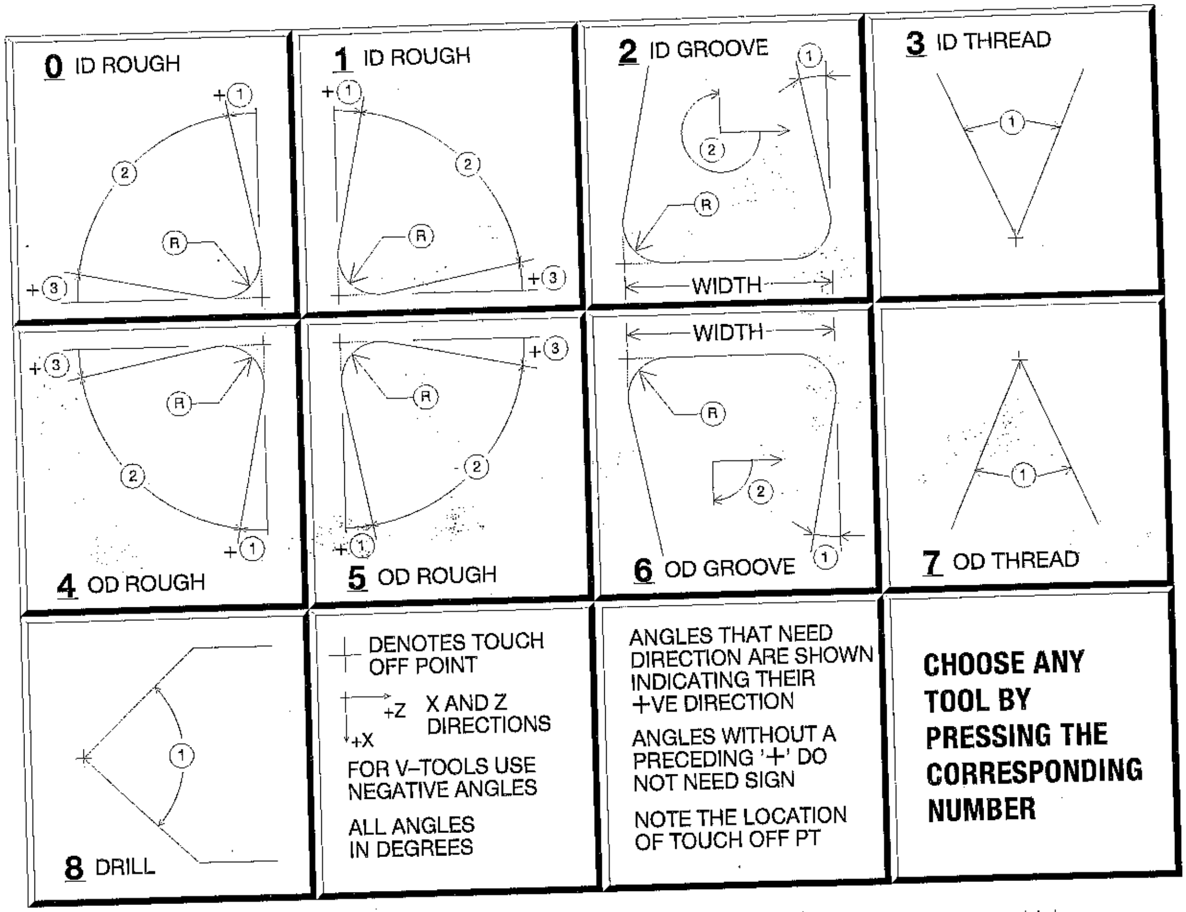


Figure 3-2 Each tool is one of eight types; when you Add a tool to the library, you will need to specify which one.

Immediately after pressing the [+ ADD] key, you will see the screen shown in Figure 3-2, which shows the 9 basic tool types used by EZPATH. After selecting the right type for the tool you'll be adding, you'll see a dialog box requesting additional information about your particular tool. Whenever you use a canned cycle in a program, EZPATH will check that the tool type entered here is appropriate for the operation you've programmed.

Be extremely careful when you enter this information: wrong information will cause your programs to operate incorrectly, and could damage your part, your tool, or the EZPATH machine! Pay special attention to the orientation and direction of angles.

For all tool types, you must specify the following information:

TOOL ID Enter a unique number for each tool. You can use any of the number keys, the "–" key, or the "." key, up to a maximum of five characters.

IN, MM Enter 0 if the settings for this tool are in inches; enter 1 if they are in millimeters.

DIST TO TOUCH OFF: X, Z For each tool, EZPATH needs to have one point that it considers the location of the tool; this is the touch-off point. If you leave these values as (0,0), the touch-off point will be where the small cross is in each tool-type picture in Figure 3-2. You can specify a different point by entering its X- and Z-coordinates. Whatever coordinates you enter will be offset values from the default touch-off point—not the tool itself. Pay extra attention to the coordinate direction display in the box after Tool Type 8, so that your use of positive and negative numbers is correct.

The other information you need to specify varies from tool to tool, and is described in the following sections.

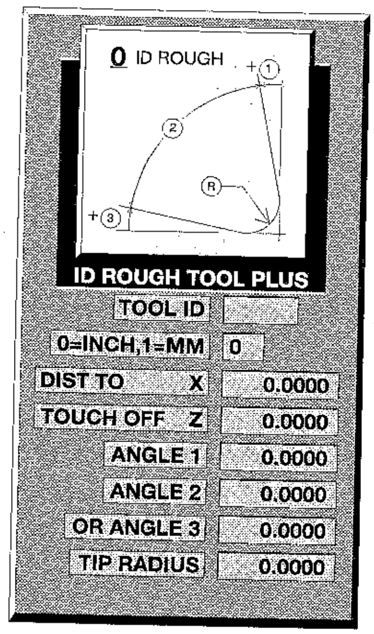


Figure 3-3 Tool type 0

3.1.1.1. Inner Diameter—Rough (Type 0)

Use this tool type for boring when the tool is moving in the positive-Z direction.

ANGLE 1 Enter the relief angle from the X-axis in the direction shown.

ANGLE 2 Enter the included angle of the tool.

ANGLE 3 Enter the relief angle from the Z-axis. Note that you only need to enter *either* angle 2 *or* angle 3; if you enter both, it will use angle 2, and disregard whatever is entered for angle 3.

TIP RADIUS Enter the radius of the tool's tip.

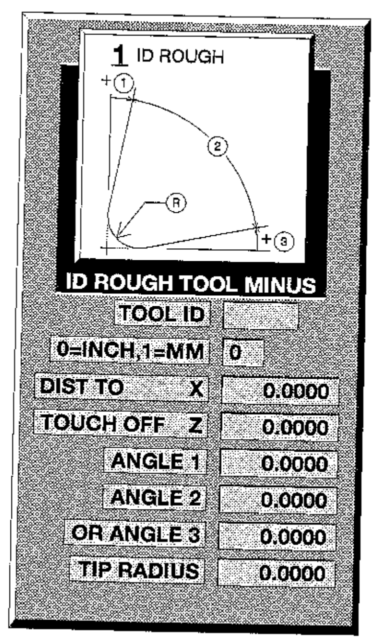


Figure 3-4 Tool type 1

3.1.1.2. Inner Diameter—Rough (Type 1)

Use this tool type for boring when the tool is moving in the negative-Z direction.

ANGLE 1 Enter the relief angle from the X-axis in the direction shown.

ANGLE 2 Enter the included angle of the tool.

ANGLE 3 Enter the relief angle from the Z-axis. Note that you only need to enter *either* angle 2 *or* angle 3; if you enter both, it will use angle 2, and disregard whatever is entered for angle 3.

TIP RADIUS Enter the radius of the tool's tip.

3.1.1.3. Inner Diameter—Groove (Type 2)

Use this tool type to cut grooves on the inner diameter of the part.

ANGLE 1 Enter the relief angle from the X-axis in the direction shown.

WIDTH Enter the *total* width of the tool.

TIP RADIUS Enter the radius of the tool's tip.

MAX DEPTH Enter the greatest depth to which the tool can cut.

GROOVE ANG 2 This is the orientation of the groove on the part—e.g. inner diameter, face, etc.

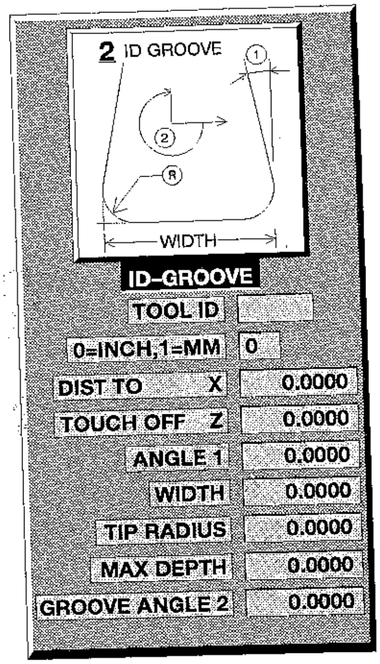


Figure 3-5 Tool type 2

A regular inner diameter groove would be 270°. Refer to Figure 3-6 for an illustration of the angle orientation.

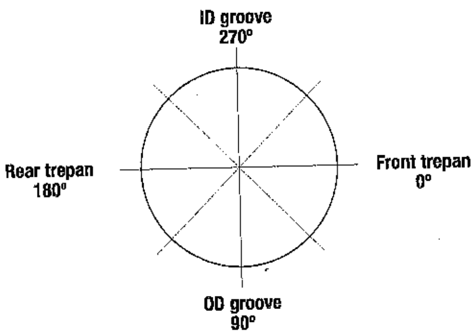


Figure 3-6 Orientation angles for grooving tools

3.1.1.4. Inner Diameter—Thread (Type 3)

Use this tool to cut threads on the inner diameter of the part.

ANGLE 1 Enter here the included angle of the tool.

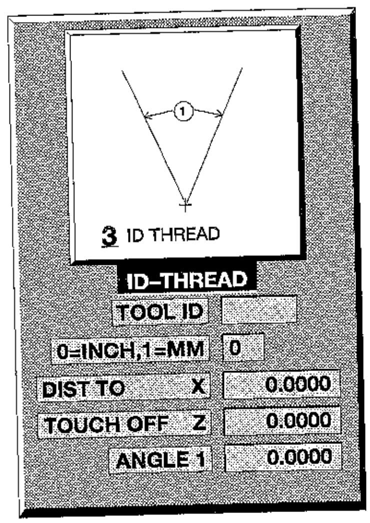


Figure 3-7 Tool type 3

3.1.1.5. Outer Diameter—Rough (Type 4)

Use this tool type for outer diameter cuts when the tool is moving in the positive-Z direction. You can use this tool for either turning or facing operations.

ANGLE 1 Enter the relief angle from the X-axis in the direction shown.

ANGLE 2 Enter the included angle of the tool.

ANGLE 3 Enter the relief angle from the Z-axis. Note that you only need to enter *either* angle 2 *or* angle 3; if you enter both, it will use angle 2, and disregard whatever is entered for angle 3.

TIP RADIUS Enter the radius of the tool's tip.

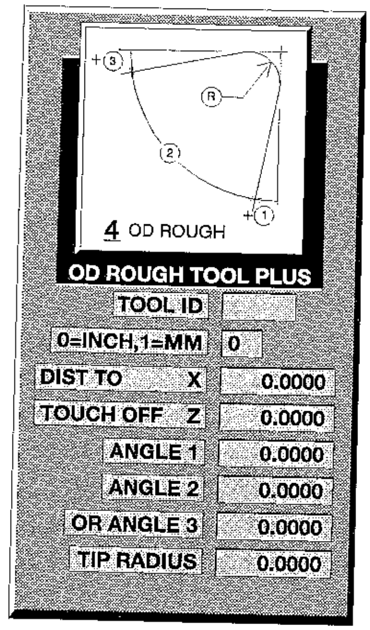


Figure 3-8 Tool type 4

3.1.1.6. Outer Diameter—Rough (Type 5)

Use this tool type for outer diameter cuts when the tool is moving in the negative-Z direction. You can use this tool for either turning or facing operations.

ANGLE 1 Enter the relief angle from the X-axis in the direction shown.

ANGLE 2 Enter the included angle of the tool.

ANGLE 3 Enter the relief angle from the Z-axis. Note that you only need to enter *either* angle 2 *or* angle 3; if you enter both, it will use angle 2, and disregard whatever is entered for angle 3.

TIP RADIUS Enter the radius of the tool's tip.

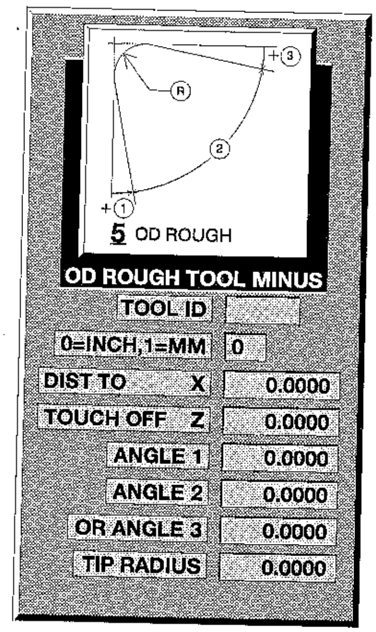


Figure 3-9 Tool type 5

3.1.1.7. Outer Diameter—Groove (Type 6)

Use this tool type to cut grooves on the outer diameter of the part.

ANGLE 1 Enter the relief angle from the X-axis in the direction shown.

WIDTH Enter the *total* width of the tool.

TIP RADIUS Enter the radius of the tool's tip.

MAX DEPTH Enter the greatest depth to which the tool can cut.

GROOVE ANG 2 This is the orientation of the groove on the part—e.g. inner diameter, face, etc. A regular outer diameter groove would be 90°. Refer to Figure 3-6 for an illustration of the angle orientation.

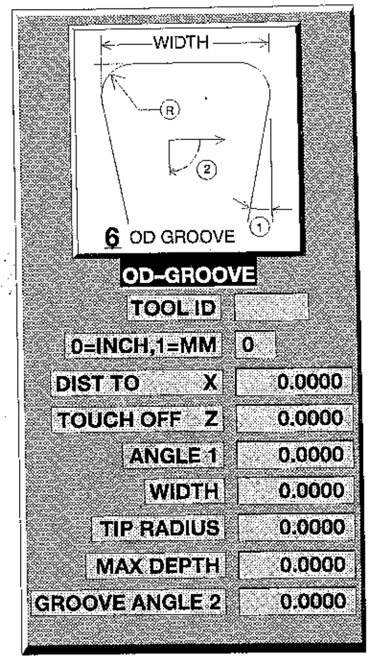


Figure 3-10 Tool type 6

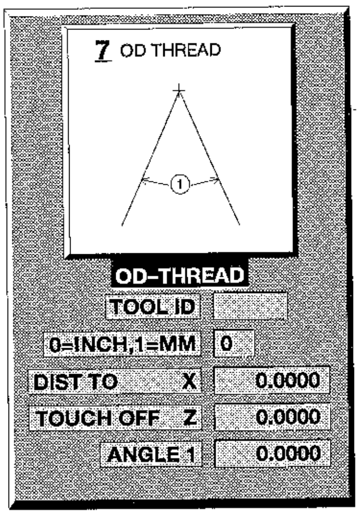


Figure 3-11 Tool type 7

3.1.1.8. Outer Diameter—Thread (Type 7)

Use this tool to cut threads on the outer diameter of the part.

ANGLE 1 Enter here the included angle of the tool.

3.1.1.9. Drill (Type 8)

Use this tool for drilling or tapping operations.

ANGLE 1 Enter here the included angle of the drill point.

TOOL DIA Enter here the diameter of the tool.

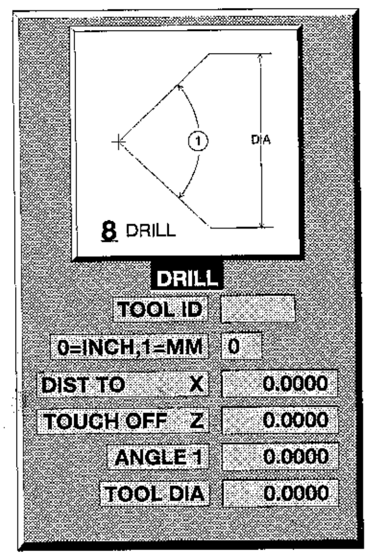


Figure 3-12 Tool type 8

3.1.2. Delete a tool

[- DEL]

Use this command to delete a tool from the library. Position the cursor on the tool to be deleted *before* selecting the Delete command.

3.1.3. Revise a tool's settings

[* REV]

Use this command to change the settings for a tool. Position the cursor on the tool you wish to revise before selecting the command; after selecting it, you will see the same data entry screen as when you add a new tool. You can change any of the tool's parameters except the tool type; to change the type, you must Add a new tool with the type you want, and then Delete the old one. Be extremely careful about changing the name: if you change the ID number, any programs which use that tool will need to be edited before they will run correctly.

3.1.4. Show the tool

[F1 SHOW]

Use this command to view a graphic representation of the tool's settings. After positioning the cursor on the tool you wish to view, press F1; a small window will appear where you will see the tool, its ID number and type, and touch-off point. It will disappear after a few seconds. Please note that the screen picture is an approximation and may not exactly match the actual tool. You can view the entire tool library at once by moving the cursor to the first line in the window (the headings line); these pictures will remain on the screen until you hit [ENTER] to clear them.

3.1.5. Save and Exit the Tool Library

[0 SAVE]

When you have finished entering and changing your tool library data, hit [0 SAVE] to save your changes and exit back to Basic Operations. To exit without saving your changes, hit [ESC].

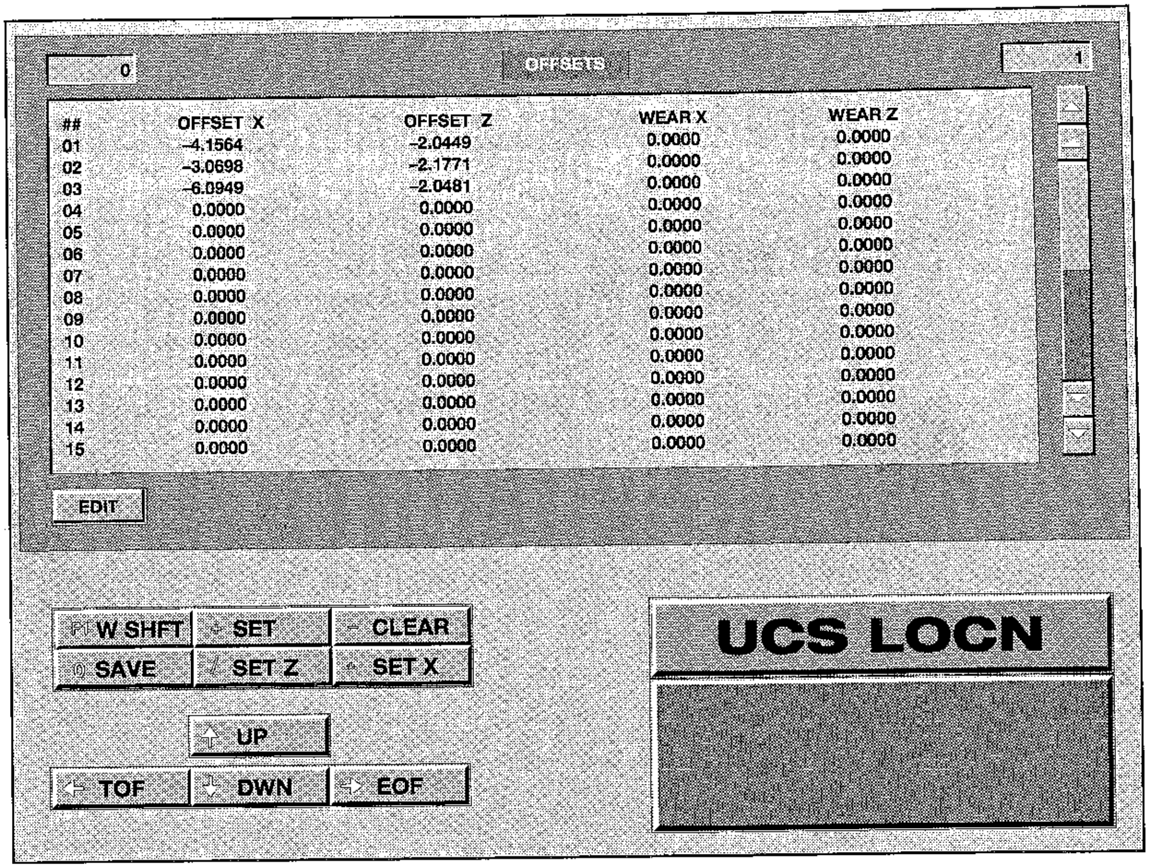


Figure 3-13 The Tool Offsets window lets you store up to 24 offsets; you can also modify any offset with wear values.

3.2. Tool Offsets

F2 T OFFS

Section 2.1.1 described how tool offsets are used to locate a particular part in the EZPATH coordinate system. An offset describes the distance from the touch-off point of a particular tool in the homed position to the origin of a part-program coordinate system. Because the offset is a measure of the relationship between a particular tool and a particular part, every time you change either the tool *or* the part, you need to adjust your offsets. Offsets are stored in a file on the EZPATH hard disk drive; you can have up to 24 offsets. Section 3.2.1 describes the different commands EZPATH makes available to you to manage your offsets data; section 3.2.2 describes how to set an offset.

Keep in mind the difference between work shifts and tool offsets as methods for defining new coordinate systems and adapting job set-ups to different parts. Tool offsets determine the origin of the Part Program Coordinate System; they are measured from the origin of the User Coordinate System. The Work Shift command *moves* the origin of the User Coordinate System. Setting a work shift of, say, –2 inches in the Z direction has the same effect as changing all of the tool offsets by that amount. Section 3.2.3 shows how you can use Work Shift to help manage multiple offsets.

3.2.1. Using the Tool Offsets window

When you first select the Tool Offsets command from the Basic Operations screen, you will see the window shown in Figure 3-13. Use the arrow keys to select an offset to work on: ↑ UP and ↓ DWN will move the cursor up and down one line, while ← TOF and → EOF will move to the top and bottom of the file. The following commands will be available:

+ SET (Set Offsets)

Use this command to enter the offset values. It will produce the dialog box shown in Figure 3-14; section 3.2.2 describes in detail how to determine the offset values.

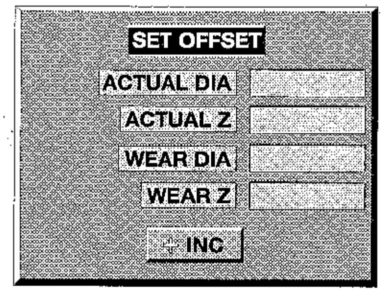


Figure 3-14 Use the Set Offset dialog box to enter your offset data.

ACTUAL DIA After making a light roughing cut with the tool, enter in this box the measured diameter of the part at the location of the cut. EZPATH will calculate the proper X-offset based on this measurement *and* its reading of the current tool location; for this reason, enter the part diameter *before* moving the tool inthe X-axis!

ACTUAL Z After making a light roughing cut on the face of the part—*before* moving the tool in the Z-axis—enter in this box what the coordinate position on the Z-axis should be. Normally, you would enter 0 here, so that the facing cut you just made would establish the Z = 0 position. Otherwise, the Z-coordinate of the current tool position will be whatever you enter here.

WEAR DIA, WEAR Z EZPATH also lets you enter wear values, so that you can account for wear of the tool—in each axis—while preserving your original offset values. Note that these are signed values: if wear has made your tool smaller, for example, enter a negative value.

By pressing the +INC key at any time, you can change to incremental data. Any data you now enter will be offset from whatever value already exists for that offset, regardless of where the tool is actually positioned. For example, to move the Z coordinate of the offset by -.0005", hit the +INC key so that you see INC Z in the dialog box, then enter -.0005 in that field.

Any values you enter in the SET OFFSET dialog box will overwrite any existing data for that offset. To revise just one value in the dialog box, hit ENTER to move through blank fields until you reach the one you want. If an entry field is blank, then the original data will remain unchanged; if there is any data in the field (even a 0), that piece of offset data will be changed accordingly.

In addition to the Set command, the following other commands are available:

/ SET Z, * SET X Use these keys as shortcuts to set just one of the above values pertaining to the offset.

- CLEAR Use this key to "zero out" the currently selected offset. If you position the cursor on the top line of the file (the headings line) and use this key, you can clear *all* of the offsets in the file.

0 SAVE Use this key to automatically save the file and exit to the Basic Operations screen. If you want to exit the Tool Offsets window without saving your work at all, use the ESC key.

F1 W SHFT Use this key to set a work shift; it functions identically to the Basic Operations work shift command described in section 2.2.8. Section 3.2.3 describes how you can use the Work Shift command together with offsets to better organize your work.

3.2.2. Setting Offsets

To set a tool offset, follow these steps:

1) Before setting an offset, use the Tool Change command (section 2.2.1) to activate it. This is not *required* for setting the offset, but it makes it easier for you to check your work.

2) After selecting F2 T OFFS from the Basic Operations screen, position the cursor over the number of the offset you wish to set.

3) Turn on the spindle, move the tool near the part, and carefully take a light roughing cut on the diameter of the workpiece. Move the tool away from the workpiece *using the Z-axis handwheel only,* without disturbing the X-axis position. Turn off the spindle and hit the * SET X key; carefully measure the diameter of the workpiece, and enter this value in the **ACTUAL DIA** box. Hit ENTER so that the Set X box is cleared, then 0 SAVE to return to Basic Operations. If the tool is active for the offset you just set, the X coordinate on the main DRO should be the value you just entered in the **ACTUAL DIA** box.

4) Turn the spindle back on, re-enter the Offsets window, and carefully face the part. Move the tool away from the workpiece *using the X-axis handwheel only,* without disturbing the Z-axis position. Turn off the spindle and hit the / SET Z key; enter 0 in the **ACTUAL Z** box. Hit ENTER so that the Set Z box is cleared, then 0 SAVE to return to Basic Operations. If the tool is active for the offset you just set, the Z coordinate on the main DRO should be the value you just entered in the **ACTUAL Z** box.

5) At this time, if you wish, you can also enter wear values for either axis as described in the previous section.

You can enter offsets for as many tools as you wish by positioning the cursor over a new offset number and repeating the above steps.

Notice that the numbers EZPATH inserts into the file, as displayed in the Tool Offsets window, do not necessarily match what you type in; EZPATH calculates the proper offset for you, based on the input you provide.

Keep in mind the difference between *setting* an offset in the Tool Offsets window, and *selecting* an offset as the active one. Just because you set a particular offset in the Tool Offsets window does not mean that it will be active when you return to the Basic Operations screen; an offset can only be activated with the Tool Change command (section 2.2.1).

3.2.3. Using Work Shift to organize offsets

If your part requires several offsets, you can use work shifts as described below to organize your offsets so that you can easily update them when your stock changes.

1) Enter the Tool Offsets window in preparation for setting the first offset. Move the cursor over offset 1, and, if there is already a value for the offset, use the [- CLEAR] key to zero it out.

2) Turn on the spindle, move the tool near the part, and carefully take a light roughing cut on the diameter of the workpiece. Move the tool away from the workpiece *using the Z-axis handwheel only,* without disturbing the X-axis position. Turn off the spindle and—instead of hitting the [* SET X] key, as you did in section 3.2.2—hit the [F1 W SHFT] key, to select the Work Shift command. Carefully measure the diameter of the workpiece, and enter this value in the **ABS DIA** box. Hit [ENTER] until the box is gone; take care not to enter anything at all, not even 0, in the **ABS Z** box.

3) Turn the spindle back on, and carefully face the part. Move the tool away from the workpiece *using the X-axis handwheel only,* without disturbing the Z-axis position. Turn off the spindle and hit the [F1 W SHFT] key again. Now, enter 0 in the **ACTUAL Z** box. Hit [ENTER] until the Work Shift box is gone.

4) Now, set the rest of your tool offsets normally, as described above in section 3.2.2. The values for offset 1 should still be 0,0.

Remember that each tool offset measures the distance from the origin of the User Coordinate System to the beginning of the part (section 2.1.2). Whenever you change the Z-position of the stock, you need to update the Z-coordinate of the offsets. What you have just done is use Work Shift to *move* the origin of the User Coordinate system to the beginning of the part—that's why, in the above steps, offset number 1 was left at 0,0. Now, if you change stock and the Z-position changes for the same program, you can face off the stock as in step 3, and use the Work Shift command to reset the Z=0 position. Since all of the offsets are meaured from (0,0) of the User Coordinate System—and this is what you are setting with Work Shift—they do not need to be further updated.

Chapter 4
Tutorial

Here in this tutorial you will be guided through the process of cutting an actual part on your EZPATH II lathe. Along the way, you'll be introduced to many of the powerful features that make it so easy to perform what used to be complicated operations.

The part you'll be cutting is illustrated in Figure 4-1. The cutting operations will take place in Do Event mode. This means that you will select a machining operation from the ones shown on the screen; enter the necessary data; and then perform that operation right away. Normally, to cut a part like this, you would write a program; in that case, you would still select the operation from the screen and enter the data, but instead of doing the operation right away, the EZPATH would create a program instruction that you could execute later.

This tutorial assumes that you have some familiarity with basic lathe operation, and also that you are familiar with the contents of chapters 1–3 of this manual—in particular, how to read the coordinate display, and how to control the tool's movement. Be sure that you have read and understand all EZPATH safety procedures before cutting any part.

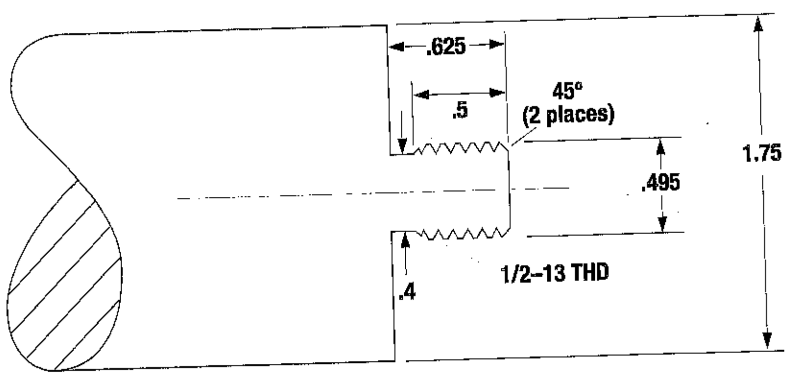


Figure 4-1 This blueprint illustrates the part that you'll be cutting in this tutorial.

4.1. Setting Up

4.1.1. System start-up

Turn on the main power switch (on the left side of the cabinet). After the software is loaded and the machine has performed its internal testing routines, you will see the message EZPATH NOT HOMED. >>HIT [MOV ABS]<<. On the lower part of the CRT screen you can see pictures of the keyboard keys with a command abbreviation on each; you can see that the MOV ABS (Move Absolute) command is on the [3 MOVABS] key. Hit this key now to select the command; then, hit the [+START] key to execute it. The tool carriage will now pull back toward you and move all the way to the right.

4.1.2. Select work material and tools

Based on the blueprint of Figure 4-1, select your work material and tools. For this part, you will need an OD turning tool (tool number 1), an OD grooving tool with a radius of .125" (tool number 2), and a 60° threading tool (tool number 3). Use a 1.75"-diameter aluminum bar for your material.

Secure your material in the chuck so that there is no less than 2" hanging out.

Now, set the spindle speed to 1060 RPM. First, set the spindle-gear lever on the front of the machine to B (12–1,180 rpm); then, from the Basic Operations screen, select Set RPM by pressing the [* RPM] key (see section 2.2.18). Type in 1060 and press [ENTER].

4.1.3. Set Tool Offsets

Tool offsets tell the EZPATH the distance from a particular tool to your part. Since each tool is a different shape and size, each tool requires a different offset. Basically, after securing your workpiece, you mount each tool and touch the part with it; EZPATH will keep track of the distance it has travelled and store it.

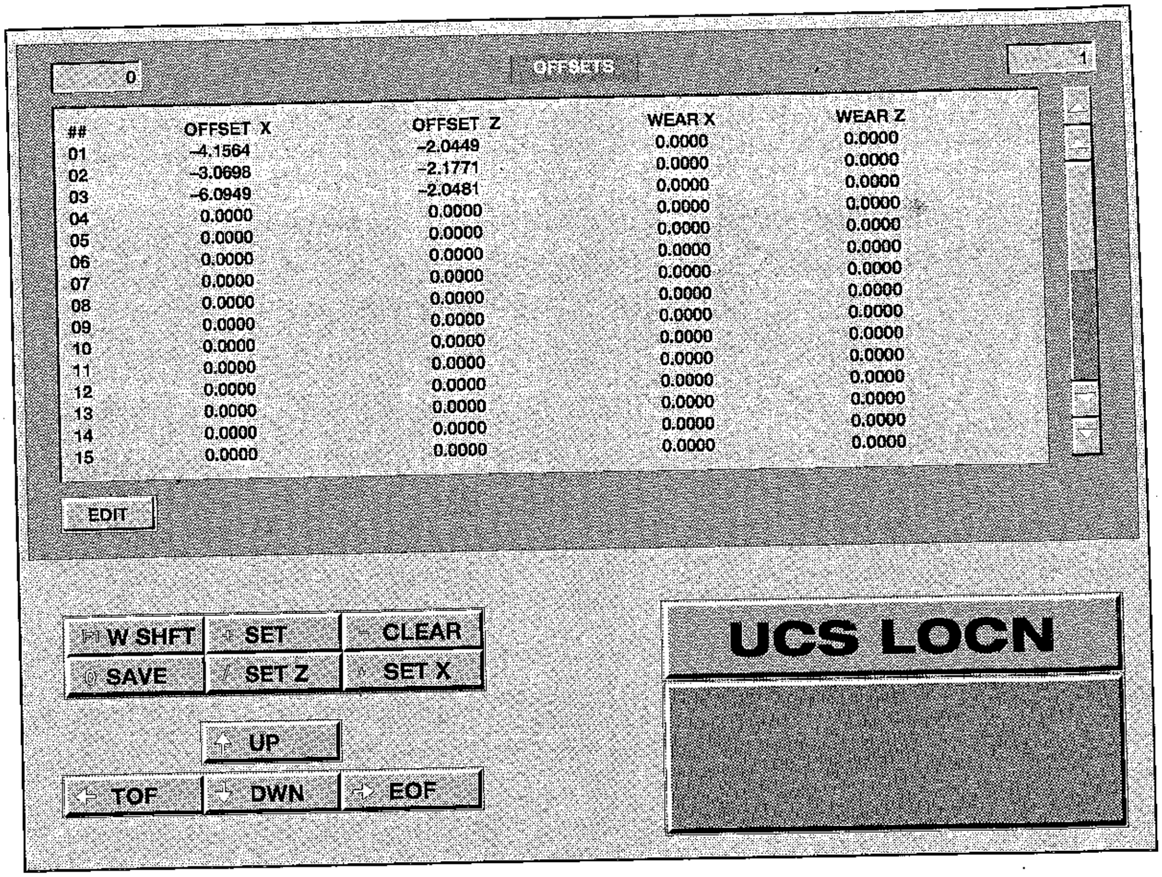
Use the following steps to set your offsets now:

1) Mount a tool in the tool holder and turn on the spindle (section 1.3.4).

2) Use the Tool Change command ([1 TL CHG]) and enter "1" for both the TOOL NO and OFFSET NO (enter "1" if this is the first tool; enter "2" and "3" for those tools).

3) Press the [F2 T OFFS] key; a screen similar to Figure 4-2 will appear.

4) Use the ↓ and ↑ keys to move the cursor over the proper offset number. Generally, you want the offset number to be the same as the tool number, so that, for example, tool



##	OFFSET X	OFFSET Z	WEAR X	WEAR Z
01	-4.1564	-2.0449	0.0000	0.0000
02	-3.0698	-2.1771	0.0000	0.0000
03	-6.0949	-2.0481	0.0000	0.0000
04	0.0000	0.0000	0.0000	0.0000
05	0.0000	0.0000	0.0000	0.0000
06	0.0000	0.0000	0.0000	0.0000
07	0.0000	0.0000	0.0000	0.0000
08	0.0000	0.0000	0.0000	0.0000
09	0.0000	0.0000	0.0000	0.0000
10	0.0000	0.0000	0.0000	0.0000
11	0.0000	0.0000	0.0000	0.0000
12	0.0000	0.0000	0.0000	0.0000
13	0.0000	0.0000	0.0000	0.0000
14	0.0000	0.0000	0.0000	0.0000
15	0.0000	0.0000	0.0000	0.0000

Figure 4-2 Use the Offsets command to tell EZPATH how to find the part with each of the tools you'll be using.

1 will use offset 1, but this is not required. If the offset number you select already has a value, that value will be overwritten.

5) Jog the tool to the workpiece, using either the handwheels or the jog knob on the control panel.

6) Manually face the workpiece a few thousandths, then turn off the spindle. Move the tool away from the part *using the X-axis handwheel only;* take care not to move the Z-axis handwheel.

7) Press the [+ SET] key; the dialog box shown in Figure 4-3 will appear. Hit [ENTER] so that the cursor is in the **ACTUAL Z** box. Enter 0; then, hit [ENTER] three times to return to the offsets screen. Use the [0 SAVE] key to exit the offsets window and return to Basic Operations; the main DRO should read **Z 0.0000**.

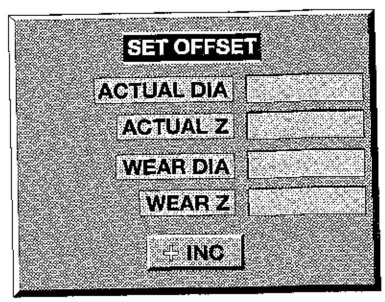


Figure 4-3 Use the Set Offsets command to input the offsets data; EZPATH automatically calculates the offset based on what you type in here.

8) Turn on the spindle again, and use the [F2 T OFFS] key to return to the tool offset window. Manually turn down the workpiece a few thousandths. Stop the spindle and move the tool away from the part *using the Z-axis handwheel only;* take care not to move the X-axis handwheel. Measure the diameter you've just cut.

9) Press the [+ SET] key again, and enter this diameter measurement in the **ACTUAL DIA** box; press [ENTER] four times to return to the offset screen. The EZPATH will use this diameter value to calculate the centerpoint of the face of your workpiece, and will make that the (0,0) point (the *origin*) whenever you use that offset number.

You can check your work by using the [0 SAVE] key to leave the offsets window and return to the main screen. When you do this, the main DRO should display an X-value identical to what you just entered.

10) Repeat steps 1–8 for each of the tools. When you have finished, the EZPATH will be able to find the exact same point with any of the tools when you select its corresponding offset number.

10) When you are done, press [0 SAVE] to save all of your new offsets. The Basic Operations screen will return to view, and you are now ready to begin cutting your part.

4.2. Roughing

4.2.1. Preparation

Figure 4-4 illustrates how your part will look after you rough out the shape using the Do Event: Rough command. You will perform 2 roughing passes. Most of the material will be removed in the first pass; then you will make a second pass to finish.

Make sure that the OD turning tool is installed in the tool holder; use the Tool Change command (the [1 TL CHG] key) to select the proper tool number and offset.

Position the tool to a suitable clearance point, outside the diameter and in the Z-positive direction (approximately X = 1.9, Z = .1). This is the point from which the tool will begin feeding toward the part.

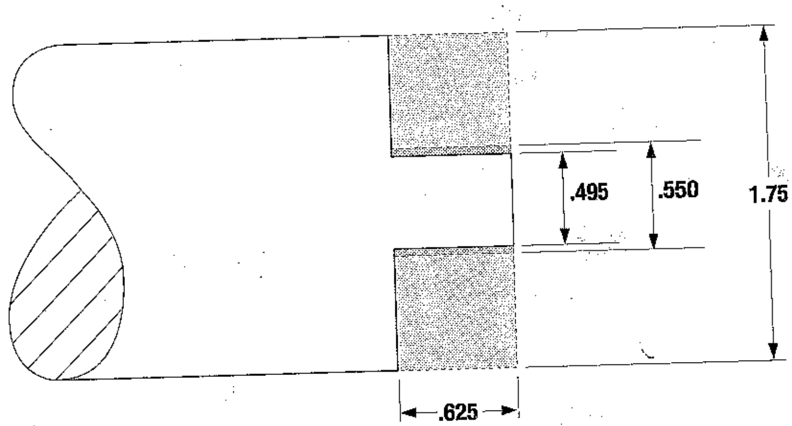


Figure 4-4 This diagram illustrates the stock that will be removed during each roughing operation.

4.2.2. Roughing the part

From the Basic Operations screen, enter Do Event mode by pressing the [2 DO EVNT] key. Then, select the Rough command by hitting the [8 ROUGH] key. You will see the dialog box shown in Figure 4-5. Enter the data as shown below:

END DIA ABS	.55 [ENTER]
END Z ABS	−.625 [ENTER]
DIA STEP	.1 [ENTER]
ENGAGE FEED	.02 [ENTER]
CROSS FEED	.01 [ENTER]
RETRACT FEED	.01 [ENTER]

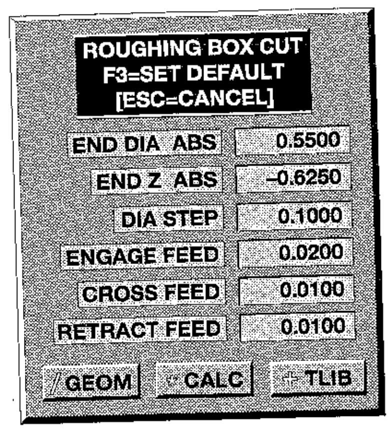


Figure 4-5 Use this data to specify the first Rough cycle.

When you hit [ENTER] the final time, the dialog box will disappear, replaced by the screen shown in Figure 4-6. You can see a command instruction which summarizes the data you have just input. As with any Do Event operation, at this time you can also use the Feed Override keys to adjust the feedrate, or the handwheels to position the tool; or, you could hit [ESC] to abort the entire operation. Use the spindle-control lever now to turn on the spindle; then, hit [+ START] to begin Roughing.

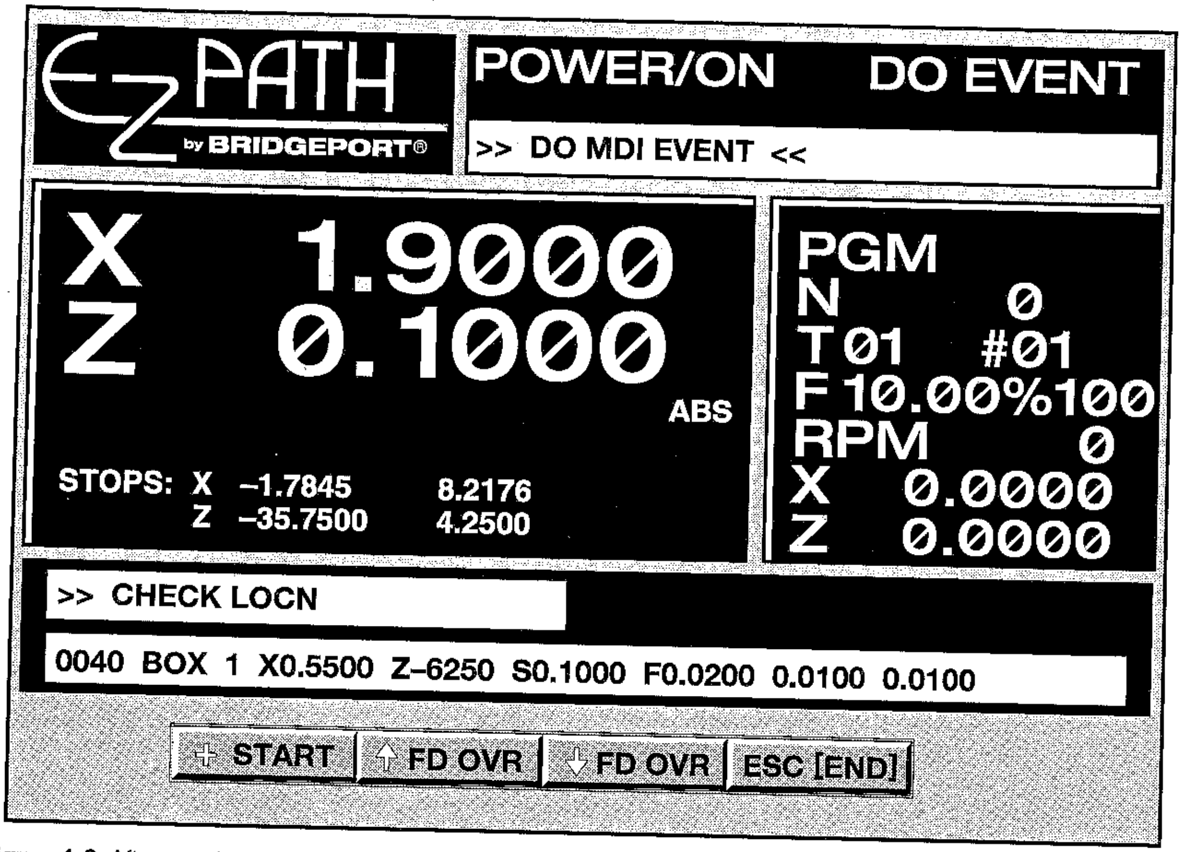


Figure 4-6 After entering all of the data, you will see this screen, where you can fine-tune the machine's status before beginning the command.

4.2.3. Finish pass

When the first Roughing operation has finished, select the Rough command again, to input the data for a finishing pass. Do not turn off the spindle or move the tool; notice that it has returned to the starting point of the previous operation.

When you see the Rough dialog box on the screen again, enter the following data:

END DIA ABS	.495 [ENTER]
END Z ABS	-.625 [ENTER]
DIA STEP	0 [ENTER]
ENGAGE FEED	.02 [ENTER]
CROSS FEED	.008 [ENTER]
RETRACT FEED	.01 [ENTER]

When this operation has finished, turn off the spindle, and hit [0 EXIT] to return to the Basic Operations screen.

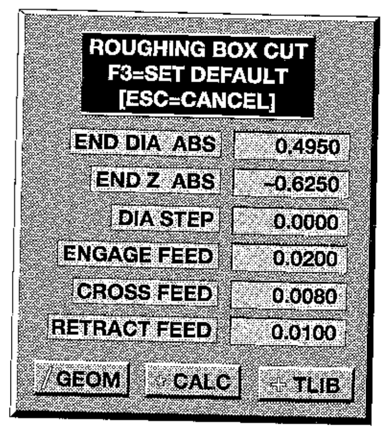


Figure 4-7 Use this data to specify the second Rough cycle, set up here as a finishing pass.

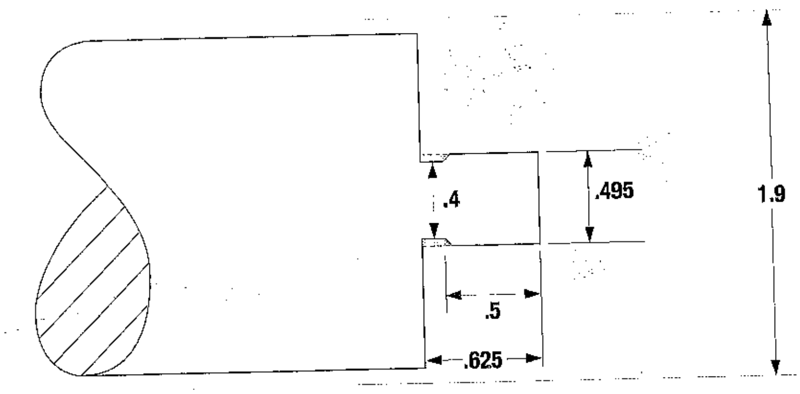


Figure 4-8 The Groove will be cut in two parts: first the groove, then the chamfer at its front face.

4.3. Groove

The next step will be cutting the groove in front of the large-diameter face. First, the groove will be cut into the part; then, the chamfer at the front of the groove will be cut as the tool moves away from the part. Figure 4-8 illustrates how the part will look after this operation is completed.

4.3.1. Preparation

First, install the OD groove tool in the holder. From the Basic Operations screen, select the Tool Change command by hitting the [1 TL CHG] key. Enter the proper tool number and offset number that you have defined for this tool.

Next, set the proper spindle speed. From the Basic Operations screen, select the Set RPM command by hitting the [* RPM] key. Type in a value of 530, and hit [ENTER].

Now, set stop values to prevent you from cutting too far into the part. Select the Stops command by hitting the [F3 STOPS] key from the Basic Operations screen. When you select the command, you will see the dialog box illustrated in Figure 4-9. Enter the data as shown below:

FAR DIA STOP	.4 [ENTER]
ACTIVATE	1 [ENTER]
NEAR DIA STOP	[ENTER] (no value entered here)
ACTIVATE	[ENTER] (no value entered here)
LEFT Z STOP	–.625 [ENTER]

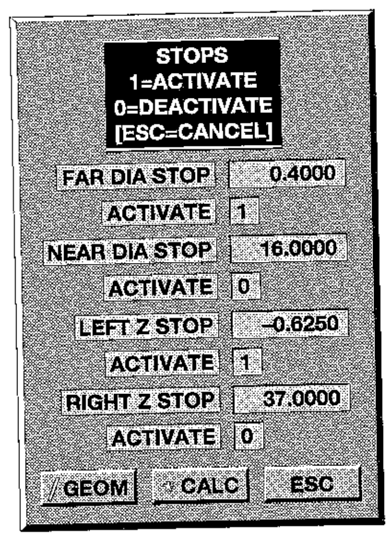


Figure 4-9 You can set stops to prevent cutting too far into the part, and to ensure that your cut stops at exactly the right place.

ACTIVATE	1 ENTER
RIGHT Z STOP	ENTER (no value entered here)
ACTIVATE	ENTER (no value entered here)

4.3.2. Cut the groove

Turn on the spindle. Then, use the handwheels or jog controls to move the tool to X=1.9, Z=–.625. Cut the groove by bringing the tool to X=.400 with the handwheel. The stop values that you have set mean that you can easily cut the groove to the precise dimensions. As you feed the tool into the part, you can see your effective feedrate on the main DRO (after the letter F).

When you have reached the bottom of the groove—without moving the tool—press 2 DO EVNT to enter Do Event mode; then, select the Chamfer command by hitting the 5 CHAMF key. You will see the dialog box illustrated in Figure 4-10. Choose type 1, and hit ENTER. Press + START to begin the chamfer.

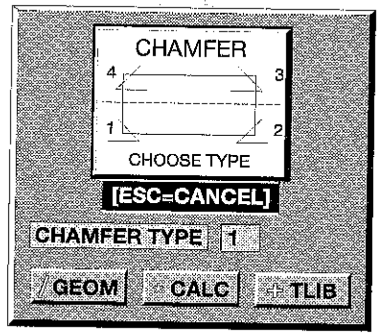


Figure 4-10 Use chamfer types 1 & 2 when the tool will approach the part from the operator side of the machine, and 3 & 4 when it will approach the part from the back.

When you hit the + START key, you will see a CHAMFER ACTIVE message on the screen. If you now rotate the Z-axis handwheel clockwise, you can see the tool moving in *both* axes in the proper proportion to cut the chamfer. When the tool has cleared the part, hit ESC to return the handwheels to normal operation. Hit 0 EXIT to return to the Basic Operations screen.

4.4. Threads

The final operation is cutting the threads. Figure 4-11 illustrates the stock to be removed by the threading operation in two steps: cutting the front chamfer, and then cutting the threads.

4.4.1. Cut front chamfer

The first step is to cut the chamfer on the front of the part. Turn off the stops by selecting the Stops command from the Basic Operations screen, and entering 0 in the ACTIVATE boxes. Then, use the handwheels or Jog mode to position the tool at X=.350, Z=.05.

Press 2 DO EVNT to enter Do Event mode, and then select the Chamfer command (5 CHAMF); choose chamfer type 2, and hit ENTER. Press the + START key to enter CHAMFER ACTIVE mode as described above.

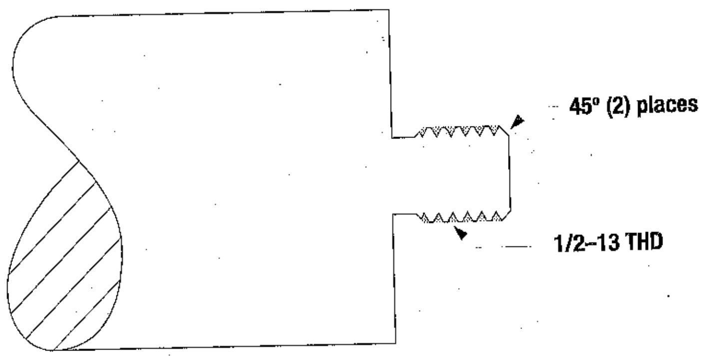


Figure 4-11 The final steps are cutting the front chamfer and then the threads.

Turn the Z-axis handwheel counterclockwise to cut the front chamfer. After the tool has cleared the part, hit the Select button on the saddle-mounted control panel. This will temporarily restore normal handwheel operation, so that you can position the tool to make a deeper cut; when you are ready, hit the Select button again to go back to CHAMFER ACTIVE mode. Repeat these steps until the proper chamfer has been achieved. Press [ESC] to return to normal operation. Press [0 EXIT] to return to the Basic Operations screen.

4.4.2. Cutting the threads

First, install the OD thread tool in the holder. From the Basic Operations screen, select the Tool Change command by hitting the [1 TL CHG] key. Enter the proper tool number and offset number that you have defined for the threading tool.

Next, use the handwheels or jog controls to position the tool to X .550, Z .20.

Press [2 DO EVNT] to enter Do Event mode, and then select the Thread command by hitting the [+ THREAD] key; you will see a dialog box like that of Figure 4-12. Enter the data shown below:

Z LEAD	.0769 [ENTER]
TARGET DIA ABS	.485 [ENTER]
TARGET Z ABS	-.6 [ENTER]

To calculate the Z-lead (the number of threads per inch) for the 1/13 thread specified in the blueprint, you can press the [* CALC] key to call up the EZPATH calculator while the cursor is positioned in the Z LEAD box. Type in 1 / 13 [ENTER] in the calculator, and EZPATH will automatically enter the answer in the Thread dialog box.

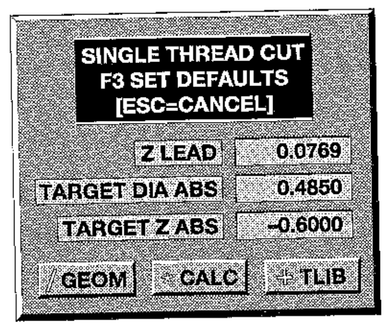


Figure 4-12 For each cutting pass, you need to select the Thread command and enter a deeper target diameter.

Turn on the spindle, then press the [+START] key to begin the threading operation. EZPATH will cut the thread in multiple passes, but for each pass, you need to select the Thread command and enter the data as above, changing the target diameter each time. Take care not to move the tool between passes. Each time you select the command, it will default to the previously used values, so you can just hit [ENTER] to skip through the **Z LEAD** and **TARGET Z** boxes. For each pass, decrease the **TARGET DIA** by .01, until you reach a target diameter of .410.

For a finish pass, enter a **TARGET DIA** of .404.

When the finish pass is completed, turn off the spindle; the part is now complete.

Chapter 5
Do Event Mode

Do Event mode is used when you wish to cut single parts or otherwise use the EZPATH II without writing a program. In Do Event mode you can perform simple positioning and cutting moves, or execute drilling, facing, turning, and threading cycles with a single command.

When you enter Do Event mode, you will see the screen shown in Figure 5-1. The >>SELECT EVENT<< prompt under the Power indicator means it is ready for you to select a command. When you select one of the commands listed on the numeric keypad, you will be prompted for the data it needs to complete the command; after you input all of the data and hit ENTER, the screen will display >>DO MDI EVENT << and the message >>CHECK LOCN<<; below this will be displayed the MDI instruction generated by the data you have just input. This line is identical to how that command would appear in a part program. At this point, you have the opportunity to turn on the spindle, verify the tool location, or otherwise prepare the machine before executing the command. Press either + START to execute the command, or ESC to cancel it.

All feedrates should be entered in inches (or mm) per spindle revolution. Note, though, that when the command is executing, the feedrate will be displayed on the screen in feet per minute (see section 1.2.4).

Many Do Event commands will let you define default values for their required data, so that the dialog boxes will already be filled in with frequently used values when you select them. If this feature is available for a particular command, the message F3 SET DEFAULTS will be displayed at the top of the dialog box; hit F3 and enter any desired values.

Remember that while a Do Event command is executing, you can stop or pause the operation with the Hold button, and then resume it with the Start button. These buttons are located on both the control unit and operator's control panel (sections 1.1.1, 1.3.3).

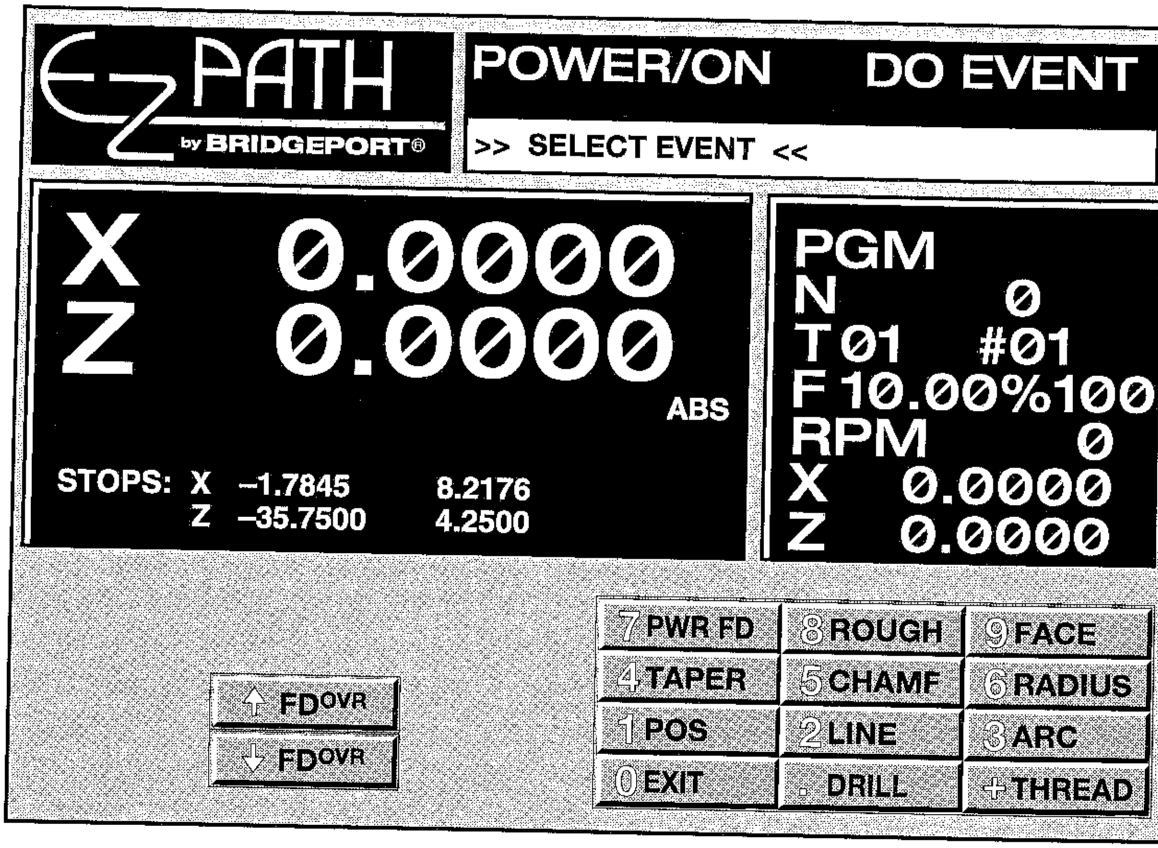


Figure 5-1 The DoEvent mode main screen; select the operations you wish to perform from the keypad at the right.

5.1. Do Event commands

5.1.1. Position

[1 POS]

Use this command to make a rapid (non-cutting) move to a specified point. In the dialog box, enter the coordinates of the destination point. **DIA ABS** and **Z ABS** mean that it is expecting absolute coordinates, relative to the (0,0) position on the screen. By pressing the [+INC] key, you can move incrementally, to a coordinate position which is measured from the current tool location. This command produces *vector* motion; i.e., the tool moves on a straight line directly to the point, not axis-by-axis.

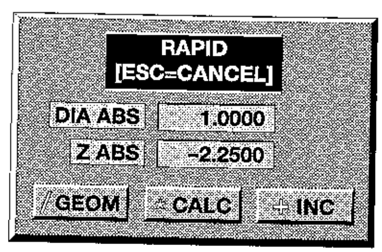


Figure 5-2 Use Position to rapid to a specific point.

The dialog box shown in Figure 5-2 will produce the following instruction:

```
0010 RAPID ABS X1.0000 Z-2.2500
```

5.1.2. Line

[2 LINE]

Use this command to cut on a line to the specified point at a specified feedrate. In the dialog box, enter the coordinates of the destination point. DIA ABS and Z ABS mean that it is expecting absolute coordinates, relative to the (0,0) position on the screen. By pressing the [+INC] key, you can move incrementally, to a coordinate position which is measured from the current tool location. In the F FEED box, enter a feedrate.

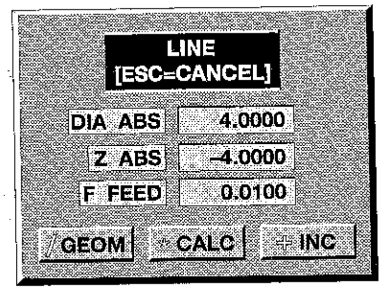


Figure 5-3 Line functions just like Position, but use it for cutting moves at a specified feedrate.

The dialog box shown in Figure 5-3 will produce the following instruction:

```
0010 LINE ABS X4.0000 Z-4.0000 F0.1000
```

5.1.3. Arc

[3 ARC]

This command is similar to Line, but it cuts an arc to the specified point, rather than a line. You also need to specify more information, as shown in Figure 5-4.

CW/CCW Specify whether you will move to the destination point in a clockwise or counterclockwise direction.

DIA ABS, Z ABS The location of the destination point in absolute coordinates. By pressing the [+INC] key, you can input incremental coordinates instead.

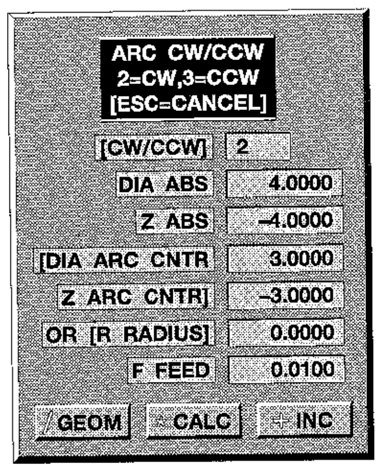


Figure 5-4 Use Arc to cut an arc of a specific size.

[DIA ARC CNTR, Z ARC CNTR] Once you input the destination point, complete the arc description by specifying *either* the centerpoint of the arc, *or* its radius. If you wish to use a centerpoint to specify your arc, enter its coordinates in these two boxes.

[R RADIUS] If you choose, instead of the centerpoint, you can specify the radius of the arc. What happens if you enter both a centerpoint and a radius? If there is *any* non-zero value in the radius field, EZPATH will ignore any centerpoint values. If you wish to use a centerpoint instead of a radius, make sure that there is 0.0000 in the radius field.

FEEDRATE Enter a feedrate here.

Like Position and Line, you can use the Arc command with incremental coordinates by pressing the [+INC] key. However, if you *do* use incremental coordinates, you must specify your arc with a radius instead of a centerpoint for the command to work properly.

The data shown in Figure 5-4 will produce the following instruction:

```
0010 ARCICNTRPT ABS CW X4.0000 Z-4.0000 XC3.0000 ZC-3.0000 F0.0500
```

(The screen display is truncated before F0.0500; it will only display as much information as will fit on one line.)

5.1.4. Taper

[4 TAPER]

Use this command to cut a taper in your part. When you select the command, you will see the dialog box shown in Figure 5-5.

First, select the type of taper you wish to cut; note that tapers 1 & 2 are for tools which approach the part from the operator side of the machine, and tapers 3 & 4 are for tools approaching the part from the rear of the machine. Then, enter the appropriate angle, measured with respect to the Z-axis as shown in Figure 5-5; this must be a positive number, measured in degrees.

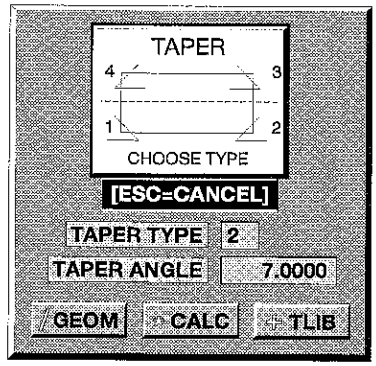


Figure 5-5 Note that the taper angle is always measured from the Z-axis, never the X-axis.

When cutting the taper, automatic feeding is disabled; instead, the X-axis handwheel is tied to the Z-axis handwheel, so that when you feed the tool into the part with the Z-axis handwheel the X-axis will automatically move in proportion to cut the taper to the specified angle. Unlike many EZPATH operations, there is no designated ending point where the tool motion will automatically stop; you must set stops appropriately to prevent cutting too far into the part (see section 2.2.16). To cut the taper, follow these steps:

1) Enter the data as shown in Figure 5-5; when you press [ENTER] after specifying the angle, it will produce the following on-screen instruction:

```
0010 TAPER T02 7.0000
```

2) Like other Do Event commands, you need to hit the [+START] key before beginning. Before hitting [+START], position the tool to where you will begin the taper. The screen

will read TAPER ACTIVE; this means that the X-axis handwheel is linked to the Z-axis as described above.

3) Use the Z-axis handwheel to make the first pass. When the tool has cleared the part—or a stop is reached—press the Select button on the apron-mounted control panel. The screen will display DUAL JOG MODE; this means that the X- and Z-axis handwheels now operate normally, and you can reposition the tool to make another pass.

4) Before making the next pass (and each successive pass), be sure to press the Select button so that TAPER ACTIVE is displayed on the screen. Be sure that your X- and Z-axis stops are set so that you do not cut or turn too far into the part.

5) After finishing your taper, press the [ESC] key; this returns the axes to normal jogging operation.

During the Taper operation, the [↑ FD OVR] and [↓ FD OVR] (Feed Override) keys can be used to scale the motion caused by the handwheel.

5.1.5. Chamfer

[5 CHAMF]

Use this command to cut a 45° chamfer in your part. When you select the command, you will see the dialog box shown in Figure 5-6. The Chamfer command functions identically to the Taper command described in the previous section, except that it automatically cuts at a 45° angle, while Taper will cut at any angle you choose.

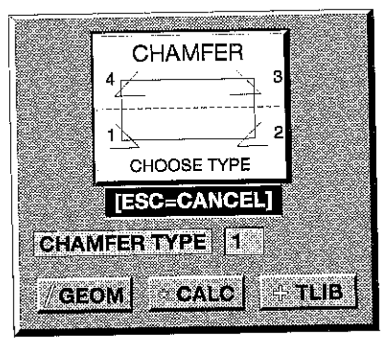


Figure 5-6 Chamfer functions just like Taper, except that you are limited to a 45° cut.

First, select the type of chamfer you wish to cut; note that chamfers 1 & 2 are for tools which approach the part from the operator side of the machine, and chamfers 3 & 4 are for tools approaching the part from the rear of the machine.

When cutting the chamfer, automatic feeding is disabled; instead, the X-axis handwheel is tied to the Z-axis handwheel, so that when you feed the tool into the part with the Z-axis handwheel the X-axis will automatically move in proportion to cut the chamfer. Unlike many EZPATH operations, there is no designated ending point where the tool motion will automatically stop; you must set stops appropriately to prevent cutting too far into the part (see section 2.2.16). To cut the chamfer, follow these steps:

1) Enter the data as shown in Figure 5-6; when you press ENTER after specifying the angle, it will produce the following on-screen instruction:

```
0010 CHAMFER T01
```

2) Like other Do Event commands, you need to hit the +START key before beginning. Before hitting +START, position the tool to where you will begin the chamfer. The screen will read CHAMFER ACTIVE; this means that the X-axis handwheel is linked to the Z-axis as described above.

3) Use the Z-axis handwheel to make the first pass. When the tool has cleared the part—or a stop is reached—press the Select button on the apron-mounted control panel. The screen will display DUAL JOG MODE; this means that the X- and Z-axis handwheels now operate normally, and you can reposition the tool to make another pass.

4) Before making the next pass (and each successive pass), be sure to press the Select button so that CHAMFER ACTIVE is displayed on the screen. Be sure that your X- and Z-axis stops are set so that you do not cut or turn too far into the part.

5) After finishing your chamfer, press the ESC key; this returns the axes to normal jogging operation.

During the Chamfer operation, the ↑ FD OVR and ↓ FD OVR (Feed Override) keys can be used to scale the motion caused by the handwheel.

5.1.6. Radius

6 RADIUS

Use this command to cut a radius, beginning at the current tool location and around a designated centerpoint. When you select the command, you will see the dialog box shown in Figure 5-7, prompting you for the centerpoint.

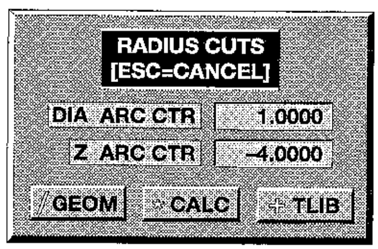


Figure 5-7 Use the Radius command to manually cut a perfectly circular radius.

When you cut a radius, tool motion in the X- and Z-axes is synchronized similar to the Taper and Chamfer commands, so that to cut a perfectly circular radius you only need to use the Z-axis handwheel. Unlike many EZPATH operations, there is no designated ending point where the tool motion will automatically stop; you must set stops appropriately to prevent cutting too far into the part (see section 2.2.16).

1) Enter the data as shown in Figure 5-7; when you press ENTER after specifying the centerpoint, it will produce the following on-screen instruction:

0010 STEPARC X1.0000 Z-4.0000

2) Like other Do Event commands, you need to hit the +START key before beginning. Before hitting +START, position the tool to where you will begin the radius. The screen will read RADIUS ACTIVE; this means that the two axes are synchronized as described above.

3) Use the Z-axis handwheel to make the first pass. When you've reached the desired point—or a stop is reached—press the Select button on the front side of the machine. The screen will display DUAL JOG MODE; this means that the handwheels now operate normally, and you can reposition the tool to make another pass.

4) Before making the next pass (and each successive pass), be sure to press the Select button so that RADIUS ACTIVE is displayed on the screen. Be sure that your stops are set so that you do not cut too far into the part.

5) After finishing your radius, press the ESC key to restore normal handwheel operation.

During the Radius operation, the ↑ FD OVR and ↓ FD OVR (Feed Override) keys can be used to scale the motion caused by the handwheel.

5.1.7. Power Feed

7 PWR FD

Use the Power Feed command to move the tool in a selected direction along one axis only. It will continue in this direction until it reaches either a stop or a machine limit. The directions (illustrated in Figure 5-8) are 2 = +X; 8 = -X; 4 = -Z; and 6 = +Z. After selecting a direction, enter a feed rate in inches per spindle revolution. To stop the tool yourself, press the Hold button, or use the the ESC key on the keypad.

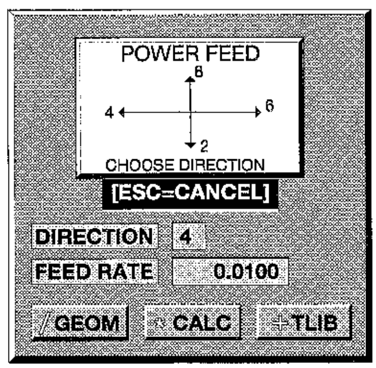


Figure 5-8 Use Power Feed to cut continuously in one direction without specifying an end point.

5.1.8. Rough

8 ROUGH

This command lets you define a roughing/box cut cycle. When you select the command, you will see the dialog box shown in Figure 5-9, which will prompt you for the following data:

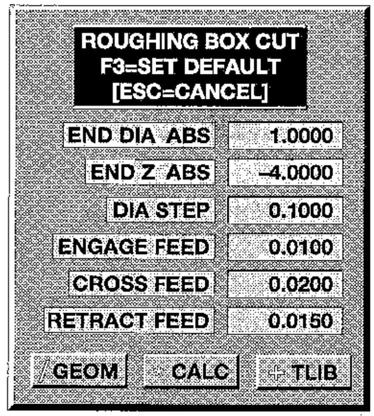


Figure 5-9 Use the Rough command to remove large amounts of material in several tool passes.

END DIA ABS, END Z ABS In these two boxes, enter the coordinates of the ending position. The amount of material removed will be the rectangle whose corners are this point and the current tool location.

DIA STEP After each crossfeed pass, the tool will advance into the part by whatever value is specified here. Note that this distance in also specified in diameter values. If you set a step value of 0, the EZPATH will perform the entire operation in 1 pass.

ENGAGE FEED This is the feedrate used when advancing in the X-axis.

CROSS FEED This is the feedrate used when advancing in the Z-axis.

RETRACT FEED This is the feedrate used when retracting after each pass. Remember that the tool is still in contact with the part at this point.

The tool motion is as follows:

1. Advance in the X-axis by the step amount at the engage feedrate;
2. Cross feed to the end Z position;
3. Retract 2 steps at the retract feedrate;
4. Rapid positioning move to the starting Z coordinate;
5. Advance in the X-axis to the next step.

The above steps are repeated until the ending diameter is reached, after which the tool withdraws to the starting position. Before executing the command, be sure to position the tool at a suitable location.

The data shown in Figure 5-9 produces the following instruction:

```
0010 BOX 1 X1.0000 Z-4.0000 S0.1000 F0.0100 0.0200 0.0150
```

5.1.9. Face

9 FACE

The Face command is identical to the Rough command described in the previous section, except that with Rough, the primary cutting action takes place parallel to the Z-axis, while with Face it is done parallel to the X-axis. Use Face when you want to achieve a smoother surface on the face of the part. Selecting Face will produce the dialog box shown in Figure 5-10, asking you for the following data:

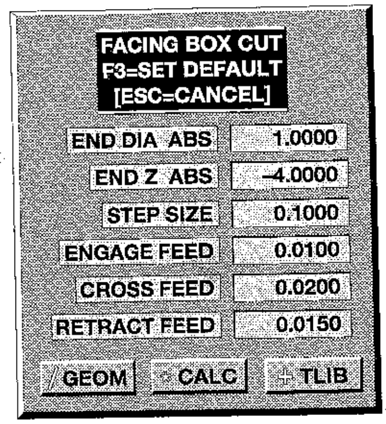


Figure 5-10 Face produces the same cutting action as Rough, but along the X-axis instead of Z.

END DIA ABS, END Z ABS In these two boxes, enter the coordinates of the ending position. The amount of material removed will be the rectangle whose corners are this point and the current tool location.

STEP SIZE After each engage feed pass, the tool will crossfeed into the part by whatever value is specified here. If you set a step value of 0, the EZPATH will perform the entire operation in 1 pass.

ENGAGE FEED This is the feedrate used when advancing in the Z-axis.

CROSS FEED This is the feedrate used when advancing in the X-axis.

RETRACT FEED This is the feedrate used when retracting after each pass. Remember that the tool is still in contact with the part at this point.

The tool motion is as follows:

1. Advance in the Z-axis by the step amount at the engage feedrate;
2. Cross feed to the end X position;
3. Retract 2 steps at the retract feedrate;
4. Rapid positioning move to the starting X coordinate;
5. Advance in the Z-axis to the next step.

The above steps are repeated until the ending Z-coordinate is reached, after which the tool withdraws to the starting position. Before executing the command, be sure to position the tool at a suitable location.

The data shown in Figure 5-10 produces the following instruction:

```
0010 BOX 2 X1.0000 Z-4.0000 S0.1000 F0.0100 0.0200 0.0150
```

5.1.10. Drill

. DRILL

The Drill command is used to drill a hole in the center of the part. You can drill the hole in pecking depths, to clear chips and prevent overheating either the tool or part. When you select Drill, you will see the dialog box shown in Figure 5-11, prompting you for the following data:

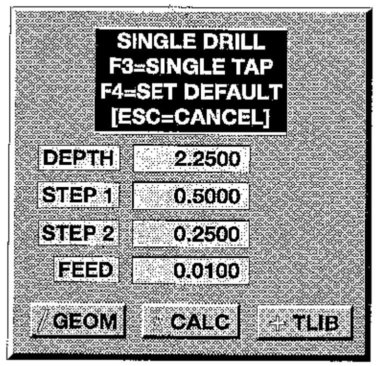


Figure 5-11 After selecting the Drill command, you can specify a Tapping operation by hitting F3.

DEPTH Enter here the total amount the tool will feed along the Z-axis. This distance represents the total distance from the current Z-coordinate position to the ending depth of the hole; meaning that it combines the depth of the hole and the distance between the tool and the part when you begin the command.

STEP 1 This is the distance between the starting tool location and the bottom of the first peck.

STEP 2 This is the distance of each successive peck.

FEED This is the feedrate for the drilling operation.

When you execute the command, the tool will first make a rapid positioning move along the X-axis to X = 0, so you do not have to position the drill along the centerline before issuing the command. Note that the X = 0 position is determined by the active offset and/or work shift; it is not "hard-wired" into the machine.

After feeding to each peck depth, the tool will perform a rapid positioning move back to the starting Z-position, and then rapid back to the previously-drilled position before resuming drilling at the feedrate. After drilling is complete, the tool will rapidly move back to wherever it was when you issued the command.

After selecting the Drill command, you can perform a Tap drilling operation by hitting F3; this is described in the next section.

The data shown in Figure 5-11 would produce the following instruction:

```
0010 DRILL D2.2500 0.5000 0.2500 F0.0100
```

5.1.11. Tap

[. DRILL] : F3 TAP

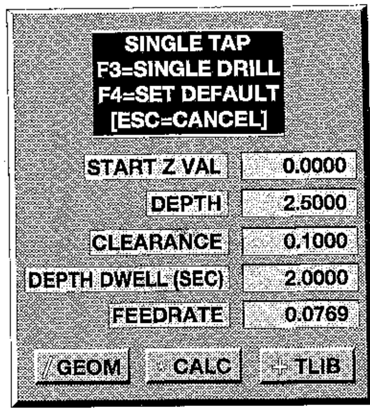


Figure 5-12 Note that by hitting F3 you can switch back to the regular Drill command.

After selecting Drill from the Do Event screen, by pressing F3 you can perform a single tap drill, where at the bottom of the hole the spindle will reverse and the tool will withdraw at the feedrate. Tap prompts you for the following data:

START Z VALUE Enter here the Z-axis position for the beginning of the hole. This is an absolute coordinate address, not an offset from the part surface.

DEPTH Enter here the total distance from the START Z VAL to the bottom of the hole.

CLEARANCE The point at which the tool will begin feeding into the part. Note that this distance is measured as an offset from the START Z position. When the operation is finished, the tool will withdraw to this clearance point.

DEPTH DWELL (SEC) After it has drilled to the required depth, but before withdrawing, the tool will dwell (i.e. not move in X or Z while the spindle rotates) for the number of seconds you specify here.

FEEDRATE This is the feedrate used for both drilling and withdrawing.

When you press the [+ START] key, the tool will rapid to the starting position, which is X = 0 and Z = the clearance plane. Note that it moves to this position on a vector, instead of separate moves in each axis. It will then drill at the feedrate to the ending depth in a single pass. The spindle will reverse direction; the tool will remain in position for the dwell time; and then withdraw at the feedrate to the starting Z position. The spindle will reverse again, and then rapid to the clearance position.

The data shown in Figure 5-12 would produce the following instruction:

```
0010 TAP Z0.0000 D2.5000 C0.1000 T2.0000 F0.0769
```

5.1.12. Thread

[→THREAD]

Use the Thread command to cut threads on either the inside or outside diameter of the part. After you select the command, it will ask you for the following data:

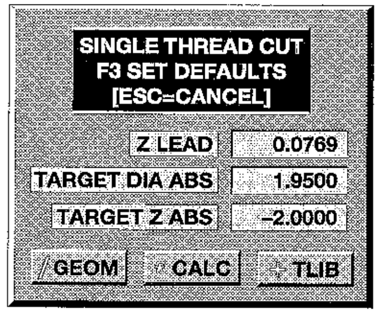


Figure 5-13 To cut threads, you need to select the Thread command for each cutting pass.

Z LEAD The lead is the distance from the top of one thread to the next. Calculate the value as 1 divided by the number of threads per inch; to cut metric threads, use the [→ MM] key in Basic Operations to place the EZPATH in metric mode before selecting this command (see section 2.2.21).

TARGET DIA ABS Use this value to define the depth of each cutting pass. Enter the absolute diameter value to which the pass will be cut. For each pass, you will need to enter a new target diameter.

TARGET Z ABS This value defines the ending Z-coordinate of the thread-cutting pass. When it reaches this point, it withdraws to the clearance plane—i.e. the X-axis position of the tool when the command was issued—and then rapids to the starting Z position.

The data shown in Figure 5-13 would produce the following instruction:

```
0010 THREAD L0.0769 X1.9500 Z-2.0000
```

5.1.13. Feed Override keys

[↑ FD OVR], [↓ FD OVR]

Use these keys to set the actual feedrate as a percentage of whatever is programmed. They work the same way here as they do in Basic Operations or Jog mode. You can specify that the actual feedrate be anywhere from 0–150% of the programmed value. Each key increases or decreases the feedrate by 5%; you can see the effect in the status window (see section 1.2.4). Any modification you set here will be in effect until it is changed, even if you do things like run a program.

Section 2.2.5 describes how Feed Override works with rapid traverse and jogging rates.

5.1.14. Exit

[0 EXIT]

Use this command to exit Do Event mode and return to the Basic Operations screen.

Chapter 6
Creating Programs

The EZPATH II includes comprehensive program development tools that let you create part programs; edit and modify them; and even preview their operation on the screen, all without leaving the EZPATH console. This chapter will describe how to create new programs in either of two ways:

- Manual Data Input (MDI) mode, where EZPATH prompts you for the information needed to create program instructions; or,
- Save Point (Teach) mode, where you can cut a part and save each point as you move to it.

In the back of this manual is the *EZPATH II Programming Workbook,* which can be reproduced for use in the shop; it walks the operator through the process of creating an MDI program step-by-step.

6.1. Using MDI

MDI programming mode is the primary method for creating new programs. Bridgeport's MDI interface makes it easy for users who have little or no CNC programming experience to create accurate programs, because once you select what type of operation you would like to program, the EZPATH machine prompts you for all of the information it needs to perform the operation; based on this information, it then automatically creates the program instruction for you. You don't need to remember any complicated commands or codes. At the same time, EZPATH includes advanced programming tools so you can create lengthy programs to machine complicated parts. Many of the Do Event commands described in the last chapter can also be programmed in MDI mode.

You enter MDI mode by pressing [8 MDI] from the Basic Operations screen. You will then see the screen shown in Figure 6-1. The large window in the center is mostly blank now, but that is where EZPATH will write the lines of program code you create. When you first enter MDI, there will already be one line in the window, which will be something similar to this:

```
0000 EZPATH SX 1 MODE I INCH FRI JAN 04 2:26:31 1996
```

This will always be the first line in any program you create. The first piece of data in this line is the line number; then, the type of machine; whether the program uses inches or millimeters; and finally, the day, date, and time the program was created. You cannot edit, move, or delete this line.

As you create program lines, EZPATH will automatically number them; they will be placed in the pro-

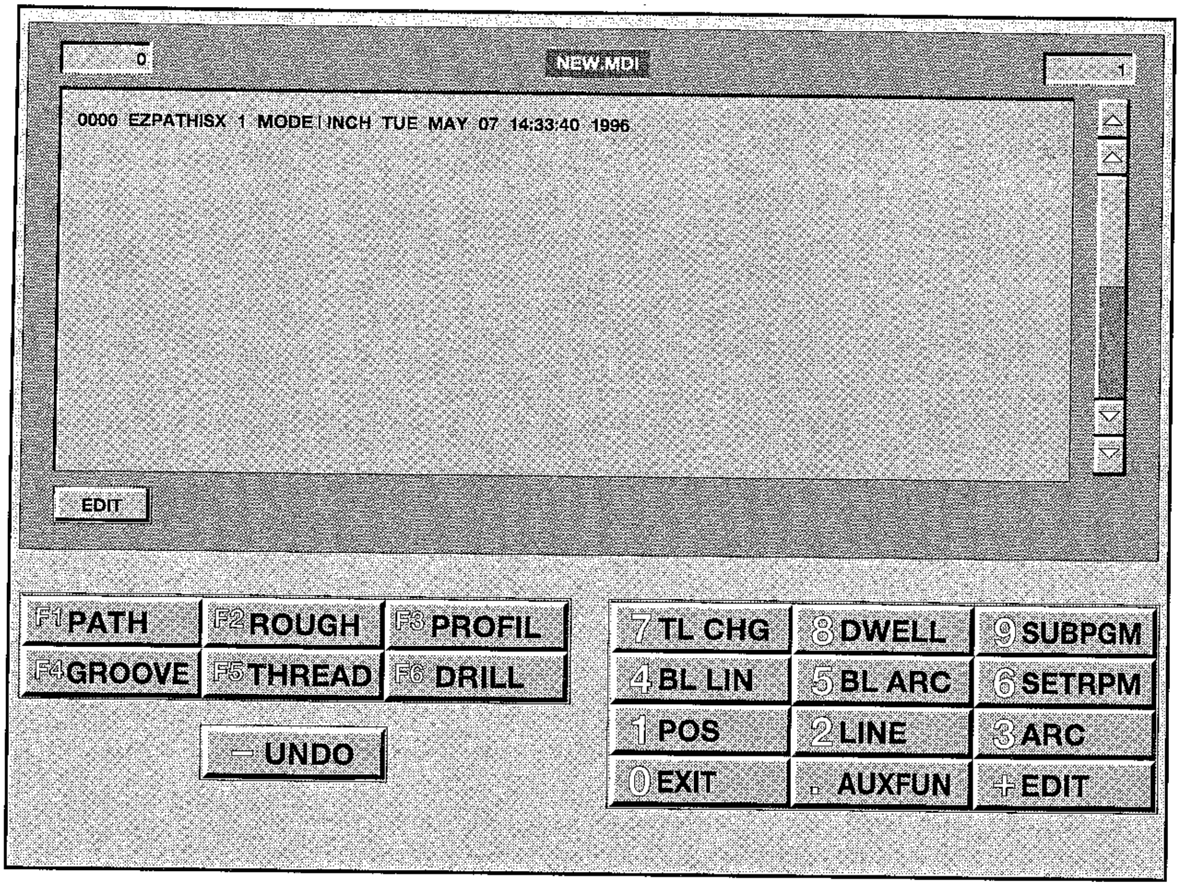


Figure 6-1 The MDI screen. MDI is the primary means for creating programs for the EZPATH; after selecting the type of cutting operation you wish to perform, the EZPATH will prompt you for all of the information it needs.

gram in the order in which you select them from the keyboard. After creating your program, you can use the EZPATH Editor to edit lines, move them around, or renumber them (see Chapter 7).

Just like in Do Event mode, any feedrates that you enter in MDI should be in inches or millimeters per spindle revolution. If you wish to program in metric units, you must place the machine in metric mode before entering MDI (section 2.2.21). With many MDI commands, after selecting the command, you can hit F2 and see a preview of the part geometry up to that point; such commands will display the **F2 VIEW PART** message in the dialog box. If you see the appropriate message in the dialog box, you can also establish default values for that command by pressing **F3 SET DEFAULTS**.

6.1.1. Edit (use EZPATH Editor)

+ EDIT

Use this command to edit your program with the EZPATH Editor. While creating your program in MDI, EZPATH enters the commands in your program in the order you select them.

The only way to make changes in MDI mode is to delete commands one by one. If you've forgotten to enter a command or have entered one incorrectly—and don't wish to delete all of the lines—you can correct your mistakes by pressing the [+ EDIT] key and entering Edit mode.

To switch back from Edit mode to MDI mode, use the Insert Line ([1 INS LN]) command. If you try to use the Exit command instead, you will be placed in Basic Operations—not back in MDI. Note: for the Insert Line command to work, the cursor must be positioned on an active line.

See Chapter 7 for a detailed discussion of the EZPATH Editor.

6.1.2. Undo last command

[- UNDO]

This command deletes the current line. If you make a mistake after you have entered a command (i.e. after EZPATH has written the program instruction in the main window), press the [- UNDO] key to delete the instruction. If you continue to press the key, it will continue to delete one program instruction each time, all the way back to the beginning of the program. Please note that there is no way to recover these lines once you have deleted them.

If you then enter new lines, it will continue to number them from the last line you entered, even if that line is now deleted. For example, if you have entered lines through 0150; press [- UNDO] to delete, say, three lines; and then enter a new line, the new line will be 0160. Use the Resequence command in the Editor (section 7.2.2) to renumber them correctly.

6.1.3. Exit MDI and Save Program

[0 EXIT]

Use this command to leave MDI mode and to save your work. After pressing the [0 EXIT] key, the cursor will be in a small window next to the word **PRGM**; here, you can enter the name of your program as it will be saved on the hard disk. You can then save your program in either of three ways:

1) Press the [+ SAVE] key to save your program and return to Basic Operations;

2) Press [* SAVE|RUN] to save and then Run it;

3) Press [/ SAVE|VIEW] to save your program and immediately preview it.

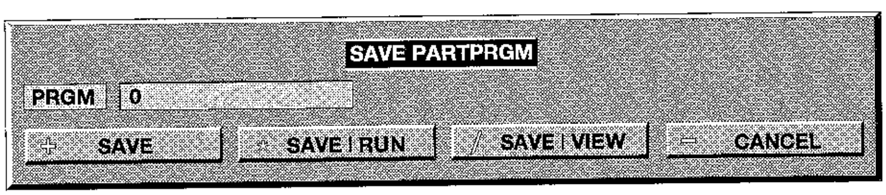


Figure 6-2 The only way to save your program is to exit MDI mode. The middle two options will also transfer you to Run mode, where you can run or preview it.

Press [- CANCEL] to return to MDI mode without saving your program; or, press [ESC] to exit MDI without saving your program.

If you have entered MDI by using the Insert Lines command from the EZPATH Editor (section 7.2.1), pressing the [+ SAVE] or [ESC] keys will return you to the Editor instead of to Basic Operations.

All programs that you create with MDI will automatically be assigned the filename extension .PGM. However, when you save your program, EZPATH automatically creates a second file called TEMP.TXT; this file contains a G-code version of your MDI program, that you can edit or modify with any text editor just like any G-code program. Every time you save a program, though, it overwrites this file; if you wish to use this file later for any reason, use the Utilities (Chapter 9) and copy TEMP.TXT right away to a floppy disk, where you can rename it.

Note that in MDI there is no other way to save your program while you are creating it.

6.2. MDI commands

6.2.1. Position

[1 POS]

The commands on keys 1–3 are basically identical to the Do Event commands described in sections 5.1.1–5.1.3.

Just like in Do Event, use the Position command to make a rapid (non-cutting) move to a specified location in the X-Z plane. The tool will move to this point at up to 450 inches per minute, so it is crucial that you verify that it will not come into contact with the part. The actual rate of motion in each axis is set with the System Parameters command (section 2.2.5); you cannot change it within the program.

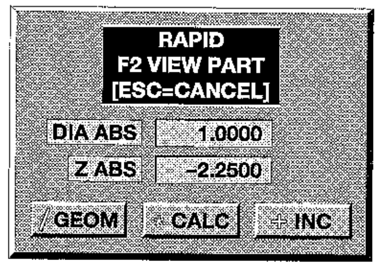


Figure 6-3 Use the Position command for non-cutting moves.

When you select the command, it will ask you to specify the destination point. **DIA ABS** and **Z ABS** mean that it expects absolute coordinates; by pressing the [+INC] key, you can input an incremental coordinate position.

The data shown in Figure 6-3 will produce the following instruction:

```
0010 RAPID ABS X1.0000 Z-2.2500
```

6.2.2. Line

2 LINE

Use this command to feed the tool on a line from the current location to a specified point. You can specify the destination location either incrementally or absolutely in the same manner as described in the Position command; use F2 to preview the part while entering the data. After entering your coordinates, it asks you to enter the F FEED value in inches (mm) per spindle revolution.

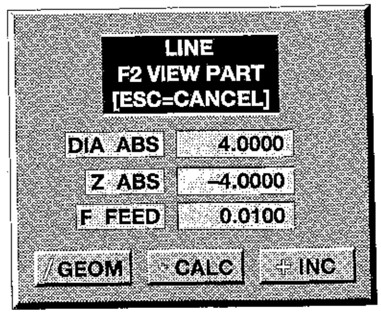


Figure 6-4 The Line command cuts a line to the specified point.

The data shown in Figure 6-4 will produce the following instruction:

```
0020 LINE ABS X4.0000 Z-4.0000 F0.0100
```

6.2.3. Arc

3 ARC

This command is similar to Line, but it cuts an arc to the specified point, rather than a line. You also need to specify more information, as shown in Figure 6-5.

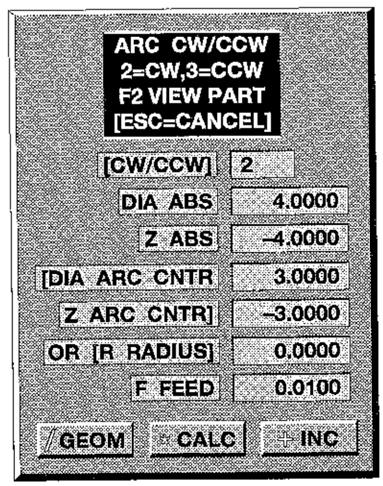


Figure 6-5 Use Arc to cut an arc from the current tool location to the specified point.

CW/CCW Specify whether you will move to the destination point in a clockwise or counterclockwise direction.

DIA ABS, Z ABS The location of the destination point in absolute coordinates. By pressing the +INC key, you can input incremental coordinates instead.

[DIA ARC CNTR, Z ARC CNTR] Once you input the destination point, you can complete the arc description by specifying *either* the centerpoint of the arc, *or* its radius. If you wish to use a centerpoint to specify your arc, enter its coordinates in these two boxes.

[R RADIUS] If you choose, instead of the centerpoint, you can specify the radius of the arc. What happens if you enter both a centerpoint and a radius? If there is *any* non-zero value in the radius field, EZPATH will ignore any centerpoint values. If you wish to use a centerpoint instead of a radius, make sure that there is 0.0000 in the radius field.

F FEED Enter the feedrate to be used in inches (mm) per spindle revolution.

Like Position and Line, you can use the Arc command with incremental coordinates by pressing the [+INC] key. However, if you *do* use incremental coordinates, you must specify your arc with a radius instead of a centerpoint for the command to work properly.

The data shown in Figure 6-5 will produce the following instruction:

```
0030 ARCICNTRPT ABS CW X4.0000 Z-4.0000 XC3.0000 ZC-3.0000 F0.0100
```

6.2.4. Blend Line

[4 BL LIN]

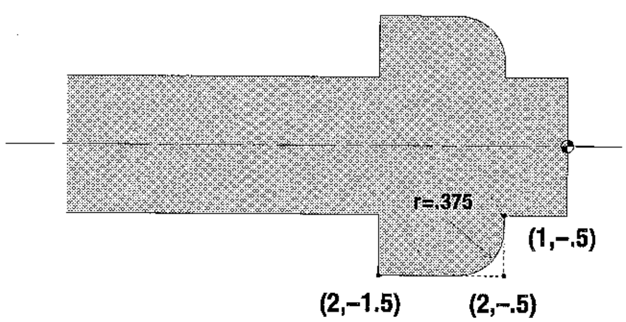
Recall that you could use the Line command to cut on a straight line. This command also does that, but before it does, it "reads ahead" to see what the next line or arc will be. It then calculates the chamfer or arc which will blend the two—*then*, it cuts the line *and* the blend arc/chamfer. It asks you for the following data:

DIA ABS, Z ABS The location of the destination point of the first line. EZPATH cuts on a line from the current point to the point you specify here, then cuts the blend arc or chamfer. However, it doesn't actually go *all* the way to the point: before it gets there, it has already begun the chamfer or arc. In Figure 6-6, you can see that, beginning from point (1,–.5), EZPATH will cut a line towards (2,–.5), which is the point specified here.

If you press the [+INC] key, you can input incremental coordinates instead of absolute ones.

CHAMF/R BLEND Enter here either the radius of the blend arc, or the dimensions of the chamfer, as appropriate. In Figure 6-6, it is the radius of the blend arc; in Figure 6-7, it is the dimension of the chamfer.

CHAMF/CW/CCW Enter 1 if you want a chamfer blend; 2 for a clockwise arc; or 3 for a counterclockwise arc.



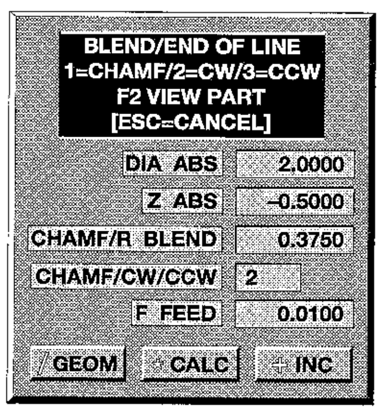


Figure 6-6 Using the Blend Line command to cut a blend arc.

F FEED Enter the feedrate to be used in inches (mm) per spindle revolution.

The program segment shown below corresponds to the data entry screen and part diagram shown in Figure 6-6. It contains three commands. Line 0020 is a line move to (1,–.5). The Blend Line command is 0030; it begins cutting a line to (2,–.5), and then cuts a blend arc of radius 0.3750. Line 0040 then cuts a line from the blend arc to (2,–1.5). The Blend Line command has already read this instruction, so the arc it has cut in 0030 will be perfectly tangent to the line created in 0040.

```
0020 LINE ABS X1.0000 Z–0.5000 F0.0100
0030 BLENDILN ABS X2.0000 Z–0.5000 R0.3750 CW F0.0100
0040 LINE ABS X2.0000 Z–1.5000 F0.0100
```

The following program lines correspond to Figure 6-7, which illustrates using the Blend Line command to blend two lines with a 0.25" chamfer. Note the differences in program line 0030—the one created by the Blend Line command.

```
0020 LINE ABS X1.0000 Z–0.5000 F0.0100
0030 CHAMFER ABS X2.0000 Z–0.5000 P0.2500 P0.2500 F0.0100
0040 LINE ABS X2.0000 Z–0.5000 F0.0100
```

In both examples, in order to cut the chamfer or blend arc EZPATH must read the information in two program lines—in these examples, 0030 and 0040. You cannot have any commands—a tool change, or program stop, for instance—between these two program lines, or EZPATH will be unable to properly execute the Blend command. This is also true of Blend Arc in the next section. The command in line 0040—the one which follows the Blend Line command—does not have to be a Line command; it could also be an Arc, another Blend Line command, or a Blend Arc.

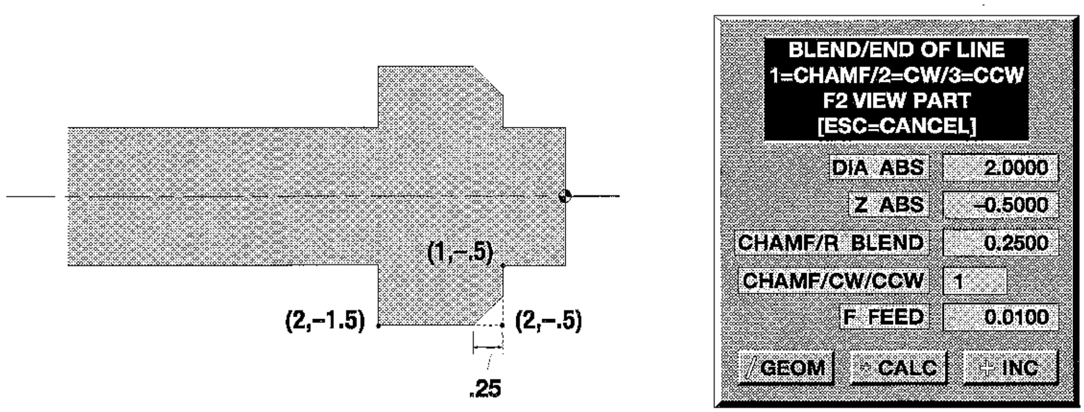


Figure 6-7 Using the Blend Line command to create a chamfer. Note that the angle of the chamfer will be automatically adjusted if the two blended lines are not perpendicular.

6.2.5. Blend Arc

5 BL ARC

This command works just like Blend Line, except that Blend Arc cuts an arc blended to a second arc, as illustrated by Figure 6-8. Arc 1 is defined by two points, plus either its centerpoint or radius. The first point on the arc is the current tool location; the second point is specified here, in the command instruction. The blend arc, which blends Arc 1 with whatever point is in the next command, is defined by its radius and direction. The Blend Arc command needs the following data:

[CW/CCW] Specify if Arc 1 will be clockwise or counterclockwise.

DIA ABS, Z ABS The location of the second point on Arc 1. By pressing the +INC key, you can input incremental coordinates, rather than absolute.

[DIA ARC CNTR, Z ARC CNTR] The centerpoint of Arc 1. Like the Arc command, you need to specify either the centerpoint *or* the radius. Do not use incremental coordinates to specify the centerpoint. Note also that if you specify a centerpoint here, for it to be effective the radius value in the next field must be 0.0000.

[R RADIUS] If you wish to define Arc 1 by its radius instead of a centerpoint, enter it here. If you are using incremental coordinates, you *must* use the radius.

R BLEND Enter the radius of the blend arc.

CW/CCW Enter 2 if you want a clockwise blend arc, or 3 for a counterclockwise one.

F FEED Enter the feedrate to be used in inches (mm) per spindle revolution.

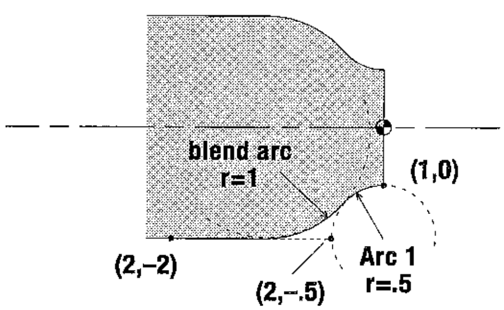
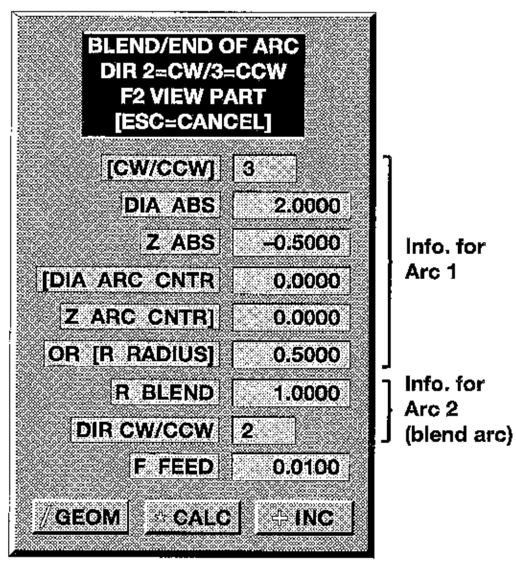


Figure 6-8 Using the Blend Arc command. See the difference between using Blend Line with a blend arc (Figure 6-6) and Blend Arc here.



The data and part diagram shown in Figure 6-8 correspond to the program instructions below. Note that the arc is specified by its radius, not its centerpoint. Line 0040 cuts on a line along the face of the part to point (1,0). Line 0050 is the Blend Arc command, which calculates an arc from (1,0) to the diameter of the line it is blending to; then calculates the blend; then cuts both arcs. Line 0060 then continues the part profile by cutting the straight line to (2,–2).

```
0040 LINE ABS X1.0000 Z0.0000 F0.0100
0050 BLENDIARCIRADIUS ABS CW X2.0000 Z–0.5000 R0.5000 R1.0000 CCW F0.0100
0060 LINE ABS X2.0000 Z–2.0000 F0.0100
```

6.2.6. Set RPM or Surface Speed

[6 SETRPM]

After you select this command, enter 1 to set a spindle rpm, or enter 2 to set a constant surface speed.

If you select 1, you will see the dialog box shown in Figure 6-9, prompting you to enter two pieces of data.

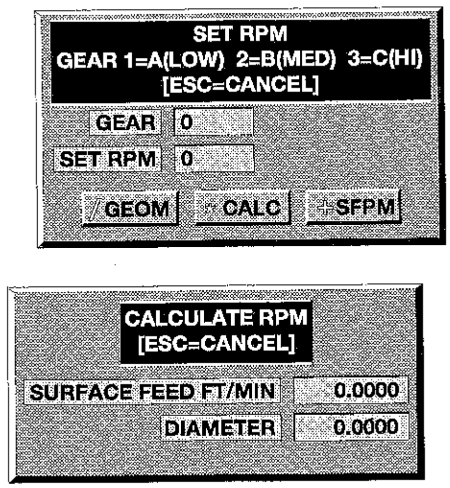


Figure 6-9 To help you calculate the desired spindle speed, you can use the Surface Speed calculator; just hit the [+SFPM] key.

GEAR Enter the number of the appropriate spindle gear. The EZPATH spindle has three gears, selected with the handle on the front of the machine (see section 1.4.1). Gear A has a range of 4–400 rpm; Gear B covers 12–1,180 rpm; and Gear C has a range of 36–3,500 rpm. Enter 1, 2 or 3 for A, B, or C, respectively.

It is important to realize that entering a value here will *not* cause the spindle gear to change. When you are running the program and this program instruction is reached, the program will stop and prompt the operator to place the machine in the gear indicated here.

SET RPM Enter the spindle speed here. Note that you can enter either a positive or negative number here, so that the spindle can rotate in either direction; use a negative value for counterclockwise, and a positive value for clockwise (a " + " sign is not necessary). You can also use a special surface speed calculator (press the [+SFPM] key) to calculate the spindle speed that will produce a particular surface speed for a particular diameter. Note that setting a spindle speed here will turn off constant surface speed, if that is the current operating mode.

The Set RPM command produces the following instruction:

```
0010 SETRPM S2500
```

If you select 2, you can program EZPATH to maintain a constant surface speed, with the dialog box shown in Figure 6-10. In this mode, the spindle speed will automatically adjust as the diameter of the workpiece changes, so that the speed of the workpiece moving past the tool tip is unchanged. You need to enter 3 pieces of information:

Figure 6-10 When using constant surface speed, make sure that the X=0 position remains the spindle centerline.

GEAR Enter a spindle gear here, as described above.

CSS VALUE Enter here the surface speed you want EZPATH to maintain, in surface-feet per minute.

MAX RPM It is important to enter a maximum spindle rpm here; as the diameter of your part decreases, the spindle speed will increase, theoretically approaching infinity as your part gets smaller. This could create dangerous operating conditions leading to serious operator injury! Enter a value here that will preserve safe operating conditions; take care also to account for the rating of the chuck you are using.

Note that if the value that has been set with the System Parameters command (section 2.2.5) is less than what you enter here, the System Parameters value will take precedence.

It is important to understand that EZPATH figures your part diameter—and hence, the spindle speed it needs—based on the X = 0 position. This position is determined entirely by the active offsets and work shifts; EZPATH does not use a "hard-wired" spindle centerline. Therefore, to use the Constant Surface Speed feature, you must have your offsets and/or work shifts set so that X = 0 corresponds to the centerline of the spindle. If you move the X = 0 position after setting CSS—either by changing the offset, a work shift, or switching to TMP mode—you will corrupt the EZPATH's spindle-speed calculations.

To turn off constant surface speed, use the Set RPM command to set the spindle to a fixed speed. A CSS command produces the following instruction:

```
0010 SETCSS G2 C200.00 S3300
```

6.2.7. Tool Change

[7 TL CHG]

The Tool Change command lets you select a new tool from the tool library and/or activate a new offset. After selecting the command, you can use the [+TLIB] key to show you all of the tools in the tool library, labeled by type and tool ID number.

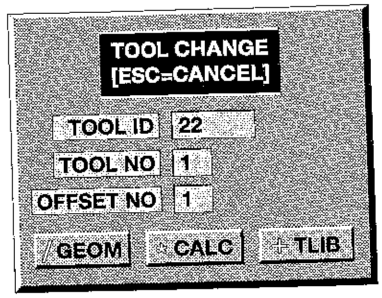


Figure 6-11 Use the Tool Change command to select a new offset, a new tool, or both.

When you select the Tool Change command, you will be asked to specify the TOOL ID, TOOL NO (tool number) and the OFFSET NO. The tool number is *not* the same as the tool ID. The "tool number" is a number that you assign to the tool based on your particular job's requirements; for example, the sequence in which tools will be used in cutting a part. In many cases, the tool number matches the offset number, so that, for example, tool number 3 uses offset number 3. The "tool ID" is used to look up information about the tool in the tool library, so that EZPATH can do things like calculate a proper cutting path for the tool. The tool ID is displayed when you hit the [+TLIB] key. The tool number is not used to get information about your tool; however, when EZPATH reaches this command in the execution of your program, it will display a message on the screen asking the operator to install the tool with whatever number you specify here. (See section 3.1 for more on the Tool Library).

Note that when you select or change an offset, it is *your responsibility* to ensure that it is the proper one. Selecting an incorrect offset can cause the tool to crash the machine or part; this could cause serious injury to you, and damage to the machine.

WARNING!

A Tool Change command produces the following program instruction; in this case, it calls for tool ID 22, which has been designated tool number 1 and offset number 1:

```
0010 TLCHG I22 T01 01
```

6.2.8. Dwell

[8 DWELL]

Use this command to insert a dwell in your program. A dwell pauses the tool in its current location for a specified period of time while the spindle continues to rotate; for this command, input a period of time in seconds. The following instruction would produce a dwell of 4 seconds:

```
0010 DWELL S4.0000
```

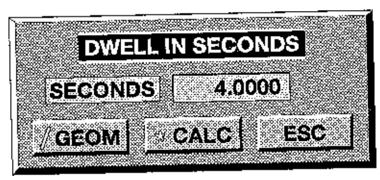


Figure 6-12 Use dwell commands to pause the tool while the spindle continues to rotate.

6.2.9. Subprogram

9 SUBPGM

You can have your EZPATH program *call* another EZPATH program. When the EZPATH machine gets to a Subprogram command in your program, it will load another program whose name you specify. It will execute all of the instructions in that program, and then go back to the program that called it. It is exactly as if you copied all of the lines from the second program to your program.

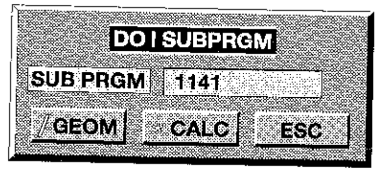


Figure 6-13 You can have your program load and execute a second program. The second program is called a "subprogram."

In order for this command to work properly, you must specify the name of a PGM file (i.e. a file whose name ends with .PGM) that is stored on the EZPATH hard disk. *You must make sure that this program is stored on the EZPATH hard disk before trying to call it with the Subprogram command.* Use the Utilities commands (Chapter 9) to verify which files are on your hard disk.

The data shown in Figure 6-13 would call the program 1141 and run it before continuing with the current program. It creates this program instruction:

```
0010  DOISUBPRGM[PGM] 1141
```

6.2.10. Auxiliary Functions

. AUXFUN

The EZPATH auxiliary functions let you control miscellaneous features of the program environment. Figure 6-14 illustrates the Auxiliary Function dialog box; simply type in the number of the desired function. The following functions are available:

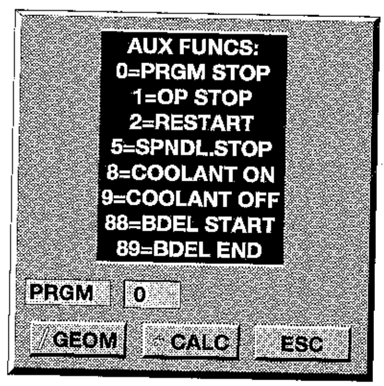


Figure 6-14 Use auxiliary functions to fine-tune your program's environment.

0: Program Stop

Stops program execution, and prompts the operator to turn off the spindle. The tool will remain in its last programmed position.

1: Optional Program Stop

This is the same as Function 0, but is executed only if the Optional Stops feature is enabled before you run the program (see section 8.1.6).

2: Program Stop & Reset

This is the same as Function 0, but it also resets the program to the beginning; otherwise, the Reset Program command must be used before the program can be run again (section 8.1.4).

5: Spindle Stop

Prompts the operator to turn off the spindle.

8, 9: Coolant

Use Function 8 to turn on the flood coolant, and Function 9 to turn it off. Coolant operation is interlocked with the spindle motor, so that when the spindle is turned off, the coolant flow automatically stops; when the spindle is restarted, the coolant flow automatically resumes.

88, 89: BDEL Start, BDEL Stop

You can use these codes to mark off sections of your program for optional execution if you have turned on the Block Delete run option (see section 8.1.6). Insert an 88 code to mark the beginning of the block, and insert an 89 code after the block; then, when the program is running, when it gets to the 88 code, it will skip over all the program instructions until it reaches the 89 code.

6.3. Canned Cycles in MDI

The next several sections introduce MDI's powerful *canned cycles*: Rough, Profile, Groove, Thread, and Drill/Tap. "Canned cycles" integrate the many different cutting actions and tool movements of complicated lathe operations in a single dialog box and a single command line—letting you specify such movements as approach and withdraw angles, different feedrates for different tool motions, finish allowances, and other aspects of your cutting operation. Most of the other cutting commands that have been described so far in this manual have required you to specify a single point or line, or a simple shape like a rectangle or radius; what makes the canned cycles so powerful is that most of them operate on a *path*: that is, you define a shape or contour that combines many lines or points, and then the canned cycle operates over the entire shape. When you select the canned cycle, you specify the path you want it to cut and then you have the opportunity to precisely specify and control every aspect of the cutting operation, tailoring it to your specific part.

Canned cycles also give you several other capabilities. When you are working with the EZPATH Editor, you can use the Verify command (section 7.2.11) to preview the tool motion of the canned cycles described here. Also, whenever a canned cycle dialog box is displayed on the screen, you can use the F1 key to get on-line help, which will describe the different entries in the dialog box. Like the other MDI commands, hitting F2 will preview the part shape to that point in the program, and F3 will let you enter default values for that particular type of cycle.

6.3.1. Define a Path

F1 PATH, 6 PTHSTP

As we just mentioned, one difference between canned cycles and other EZPATH operations is that most canned cycles operate on a *path*. A path lets you combine many simple commands (Line, Arc, etc.) to define a complete shape or profile, and then assign to it a unique name. When you select a canned cycle, you specify the path name instead of just a single destination point. The cutting operation of the canned cycle will then take place on all of the points in the path. You can also select the same path for different operations—say, Rough and then Profile—so that you only need to define it once.

A path definition has three parts:

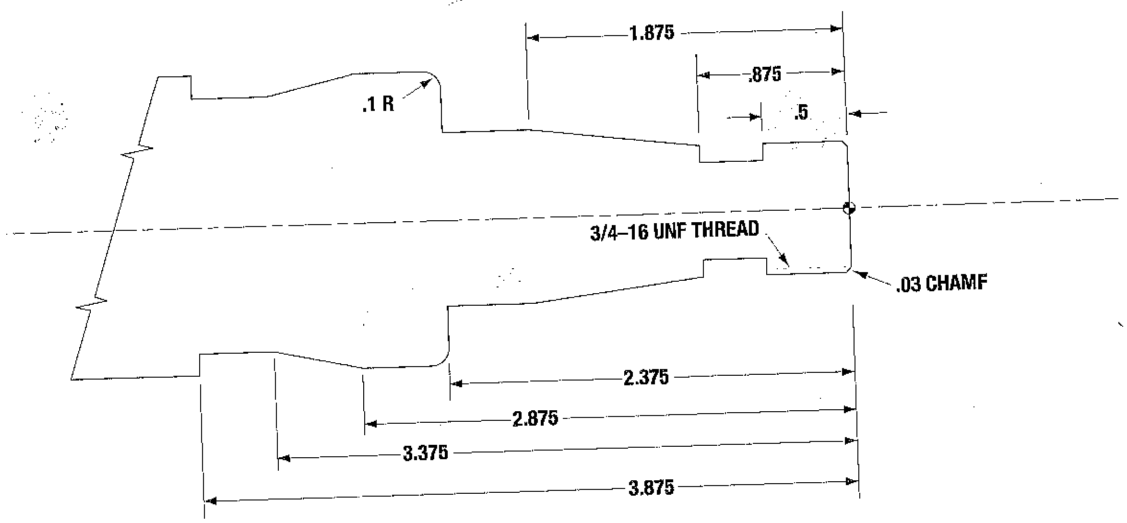
1) The first line in the path definition is the **STARTPATH** command, which is created when you select the Path command (F1 PATH key). This command is also where you enter the name of the path; later, when you select your canned cycle(s), it will refer to the path by this name. In the sample program below, these commands are in line 0010 and 0130.

Note that each path you define must have a unique name. If your program uses subprograms (section 6.2.9), be sure that any path names in the subprogram do not duplicate path names in the main program.

2) The next part of the path is the various commands which actually define the part shape. After entering the name of the path, you build the path shape with Line and Arc commands, together with Blend Line and Blend Arc. These commands all function the same as they normally do in MDI (see 6.2.1–6.2.5), except that any feedrates are ignored (these will be specified when you select your canned cycle). The first command in each path (as in line 0020 below) must be a rapid positioning move to the first point of the path; after this initial move, do not use the Position command in your path definition. While you are entering your path, you can use the - UNDO key to erase mistakes; it works just like it always does in MDI (see section 6.1.2).

3) The third part of the path definition is the **PTHSTP** instruction, which is created when you select the Path Stop command by hitting the 6 PTHSTP key. (Note that this key is only displayed while you are entering a path.) When you select the Path Stop command, EZPATH will ask you to type in a "0" to confirm. Path Stop produces lines 0120 and 0180 in the above listing.

The program listing reproduced below will cut the part whose blueprint is illustrated in Figure 6-15; all of the canned cycles in the following sections (except Drill) will refer to this part and program listing. In this listing, all of the commands not directly pertaining to the Path command have been blocked in gray, so that you can clearly see the Path definitions.



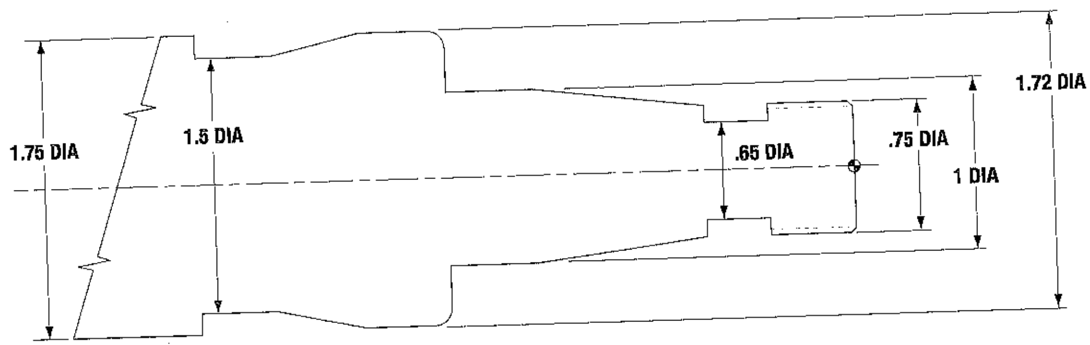


Figure 6-15 The sample part used in the MDI canned cycle examples. Path 1 describes all of the contour information except the innermost groove, which is described in Path 2.

```
0000 EZPATHISX 1 MODEIINCH SUN JUL 07 20:08:05 1996
0010 STARTPATH 1
0020 RAPID ABS X0.0000 Z0.0000
0030 CHAMFER ABS X0.7500 Z0.0000 P0.0300 P0.0300 F0.0100
0040 LINE ABS X0.7500 Z-0.8750 F0.0100
0050 LINE ABS X1.0000 Z-1.8750 F0.0100
0060 LINE ABS X1.0000 Z-2.3750 F0.0100
0070 BLENDILN ABS X1.7200 Z-2.3750 R0.1000 CW F0.0100
0080 LINE ABS X1.7200 Z-2.8750 F0.0100
0090 LINE ABS X1.5000 Z-3.3750 F0.0100
0100 LINE ABS X1.5000 Z-3.8750 F0.0100
0110 LINE ABS X2.0000 Z-3.8750 F0.0100
0120 PATHSTOP
```

```
0130 STARTPATH 2
0140 RAPID ABS X0.7500 Z-0.5000
0150 LINE ABS X0.6500 Z-0.5000 F0.0100
0160 LINE ABS X0.6500 Z-0.8750 F0.0100
0170 LINE ABS X0.7500 Z-0.8750 F0.0100
0180 PATHSTOP
0190 RAPID ABS X2.0000 Z3.0000
0200 TLCHG I11 T01 01
0210 SETCSS G3 C675.00 S3200
0220 AUXFUN M8
0230 ROUGH 1 I1 X0.0100 Z0.0100 F0.0100 0.0100 0.0100 S0.1000 C0.1250 W45.0000 W0.0500 D2 U1 A1
0240 RAPID ABS X2.0000 Z3.0000
0250 SETRPM G3 S850
0260 TLCHG I93 T02 02
0270 GROOVE 2 3 A0.0030 F0.0100 R0.0100 P0.0000 C0.0130 O80.0000 L0.0500 D1.0000
0280 RAPID ABS X2.0000 Z3.0000
0290 TLCHG I60 T03 03
0300 SETRPM G3 S530
0310 THREAD 1 L0.062500 H0.0383 S0.0050 0.0030 0.0020 #2 C0.1000 Z0.1500 -0.5500 D0.7500 0.7500
   29.0000
0320 RAPID ABS X2.0000 Z3.0000
0330 AUXFUN M2
```

The listing above includes two path definitions, named 1 and 2 (lines 0010 and 0130). Path 1 includes all of the part shape except the groove behind the threaded section; this path is used by Rough and Profile. The groove is defined in Path 2, which is used by a Groove command. Note that the EZPATH does not begin to actually cut the part until line 0230, when the Rough command is issued and calls Path 1 for the first time.

Generally speaking, path definitions are placed at the beginning of the program, although this is not required. The path definition, though, must come before the command which calls it.

The *Programming Workbook* at the back of this manual provides a step-by-step guide to the creation of the sample program listed here.

6.3.2. Rough

F2 ROUGH

The Rough command here in MDI is a more comprehensive version of the Rough command in Do Event. There are a number of key differences: Do Event Rough produced basically a rectangular cut-out, whereas in MDI at the end of each pass the tool will follow the contour programmed in the path. Also, you can precisely specify each aspect of the tool motion, including feedrates for each type of cutting move, clearance positions, etc. Note that values

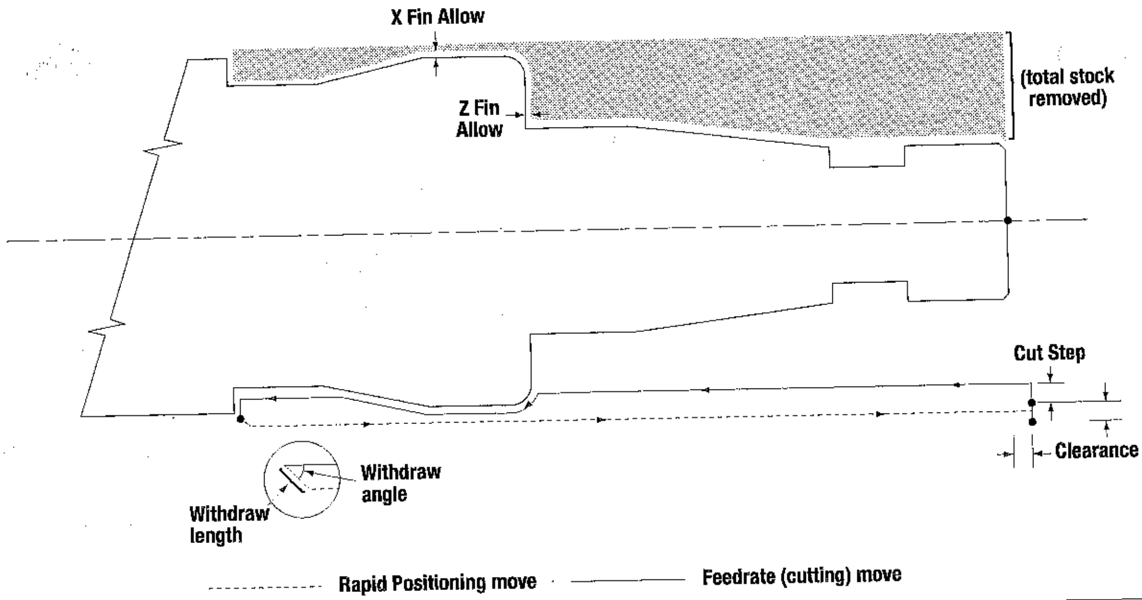
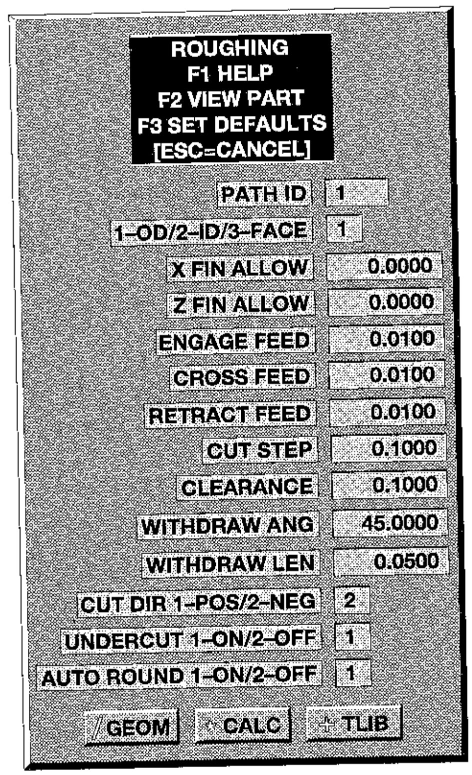


Figure 6-16 The above diagram illustrates a cutting pass produced by the Rough cycle, operating on Path 1 of the sample program. Some of the tool motions are exaggerated for clarity's sake.

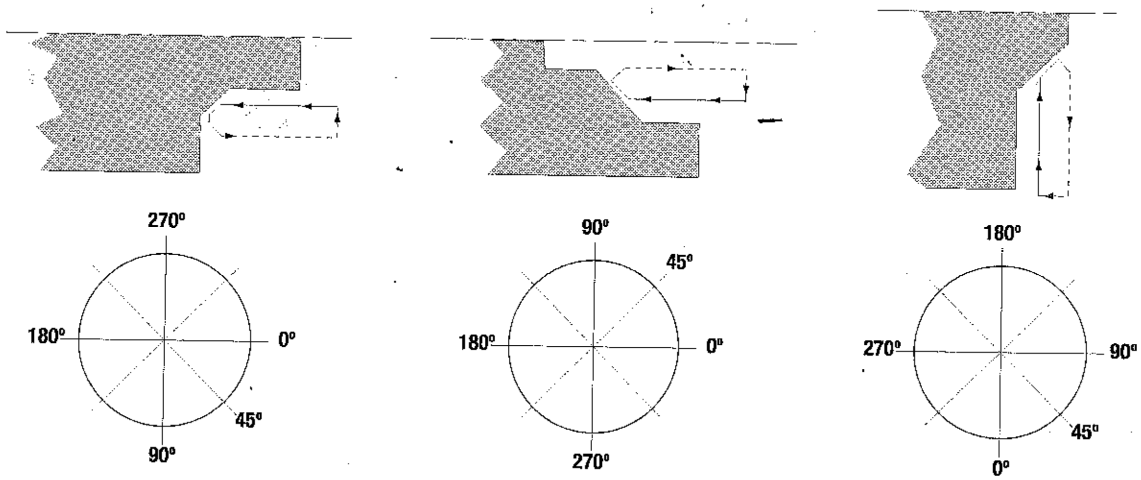


for finish allowance, cut step, and clearance—as they pertain to the X-axis—are *not* diameter values. When you select the command, a dialog box like in Figure 6-16 will prompt you for the following data:

PATH ID Enter here the name of a previously-defined path.

1-OD/2-ID/3-FACE Enter 1 if the roughing cycle will cut on the outer diameter; 2 for the inner diameter; or 3 for a facing cut. The facing cycle resembles the Face command in Do Event, in that the major cutting action is along the X-axis.

X FIN ALLOW Enter here the amount of material that will be left on the diameter of the part, to be removed in a finishing pass. If the programmed diameter of the part is 1", and the finish allowance is .05", then the Roughing cycle will only cut to diameter 1.1". (Since the finish allowance is not a diameter amount, the diameter of the unfinished part is 1.1", not 1.05".)



Type of rough cycle: 1–OD
Cut direction: 2–Neg
Withdraw angle: 45°

Type of rough cycle: 2–ID
Cut direction: 2–Neg
Withdraw angle: 45°

Type of rough cycle: 3–Face
Cut direction: 2–Neg
Withdraw angle: 45°

Figure 6-17 Depending on the type of roughing cycle (OD, ID, Face) and cutting direction (positive or negative), the cutting action can be oriented six different ways. Each type is illustrated above with a part fragment; for each, you can see the cut step; the crossfeed move; the withdraw move; and the rapid move back out to the clearance plane. The "wheel" under each diagram shows you the orientation for the withdraw angle.

Z FIN ALLOW Enter here the amount of material that will be left on the part in the Z-axis direction, to be removed in a finishing operation.

ENGAGE FEED This is the feedrate that will be used for any cutting moves in which the tool is approaching the part centerline.

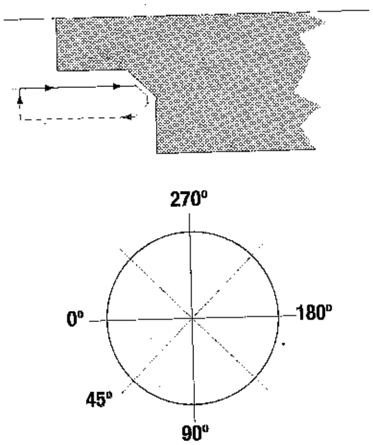
CROSS FEED This is the feedrate that will be used for straight Z-axis cuts (X-axis, for facing cycles).

RETRACT FEED This is the feedrate that will be used for cutting moves away from the part centerline.

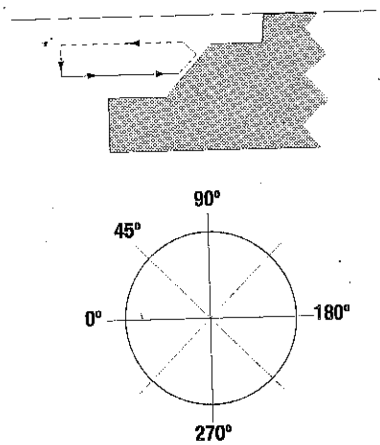
CUT STEP This is the depth of each cutting pass. For an OD rough, this is an X– move; for an ID rough, it is an X+ move; and for a facing cycle, it is a Z– move.

CLEARANCE This is the distance from the part in each axis to which the tool will move at the rapid traverse rate before the Rough cycle begins. The tool will also retract to this position after each cutting pass.

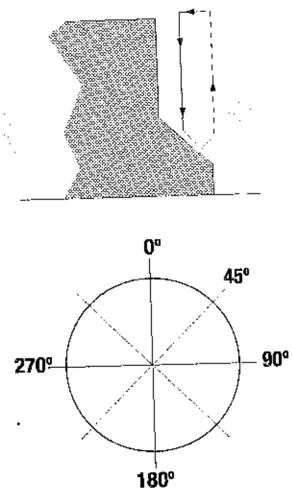
WITHDRAW ANG After the tool has reached the finish allowance position on each pass, it will withdraw from the part at this angle. Measure the angle based on the direction of



Type of rough cycle: 1–OD
Cut direction: 1–Pos
Withdraw angle: 45°



Type of rough cycle: 2–ID
Cut direction: 1–Pos
Withdraw angle: 45°



Type of rough cycle: 3–Face
Cut direction: 1–Pos
Withdraw angle: 45°

the main crossfeed moves. Figure 6-17 illustrates this for each type of roughing operation. The withdraw move takes place at whatever feedrate you specify in RETRACT FEED.

WITHDRAW LEN This is the length of the move the tool makes at the withdraw angle after each cutting pass.

CUT DIR 1-POS/2-NEG For OD roughing (type 1) and ID roughing (type 2), the cutting direction is negative if it is toward the headstock, positive if toward the tailstock—corresponding to the orientation of the Z-axis. For facing cycles (type 3), the cutting direction is the direction along the X-axis. The cutting direction you enter here should correspond to the tool type.

UNDERCUT 1-ON/2-OFF The EZPATH uses an undercut checking feature. When it is turned on, the EZPATH will check your program for undercuts when it loads; if one is found, it will tell you. Enter a 2 here to disable this feature.

AUTOROUND 1-ON/2-OFF The EZPATH also normally uses an autoround feature when calculating the path of the tool. This means that, when a sharp corner is programmed in the part, the path of the tool will have built into it a small arc at the corner equal to the radius of the tool, such that the tool will always remain in contact with the part and overall tool movement is minimized. The shape of the part is unaffected by this. If this is not desired, enter a 2 here. Figure 6-19 illustrates the effect.

Figure 6-16 outlines the tool motion produced by the Rough cycle. When the command is first called, the tool rapids to the clearance point; if you draw a line out from the first and last points defined in the path, and then add the clearance amount in each axis, you will have the clearance point. Next, it will move towards the part at the engage feed rate and cut into the part for the cut step amount.

It then crossfeeds in the cut direction. At the end of each crossfeed move, it will cut back for the cut-step amount. Whenever it meets the programmed part shape, it will cut around it.

At the end of each cutting pass, it moves away from the part at the withdraw angle at the retract feed rate. It then rapids away from the part to the clearance plane.

The depth of the next cutting pass is the start point of the previous pass plus the cut-step amount; it is not affected by the withdraw length.

Depending on the type of roughing cycle you select and the cutting direction, the cutting motions can be oriented in one of six different ways: OD+, OD–, ID+, ID–, Face+, and Face–. Figure 6-17 shows you a simple cutting pass for each, so you can see the difference.

The following program listing—excerpted from section 6.3.1—shows a path definition and the Rough instruction which calls it. The data elements in the Rough instruction (line 0230) are in the same order as entered in the dialog box of Figure 6-16.

```
0010 STARTPATH 1
0020 RAPID ABS X0.0000 Z0.0000
0030 CHAMFER ABS X0.7500 Z0.0000 P0.0300 P0.0300 F0.0100
0040 LINE ABS X0.7500 Z-0.8750 F0.0100
0050 LINE ABS X1.0000 Z-1.8750 F0.0100
0060 LINE ABS X1.0000 Z-2.3750 F0.0100
0070 BLENDILN ABS X1.7200 Z-2.3750 R0.1000 CW F0.0100
0080 LINE ABS X1.7200 Z-2.8750 F0.0100
0090 LINE ABS X1.5000 Z-3.3750 F0.0100
0100 LINE ABS X1.5000 Z-3.8750 F0.0100
0110 LINE ABS X2.0000 Z-3.8750 F0.0100
0120 PATHSTOP
0130 STARTPATH 2
0140 RAPID ABS X0.7500 Z-0.5000
0150 LINE ABS X0.6500 Z-0.5000 F0.0100
0160 LINE ABS X0.6500 Z-0.8750 F0.0100
0170 LINE ABS X0.7500 Z-0.8750 F0.0100
0180 PATHSTOP
0190 RAPID ABS X2.0000 Z3.0000
0200 TLCHG I11 T01 01
0210 SETCSS G3 C675.00 S3200
```

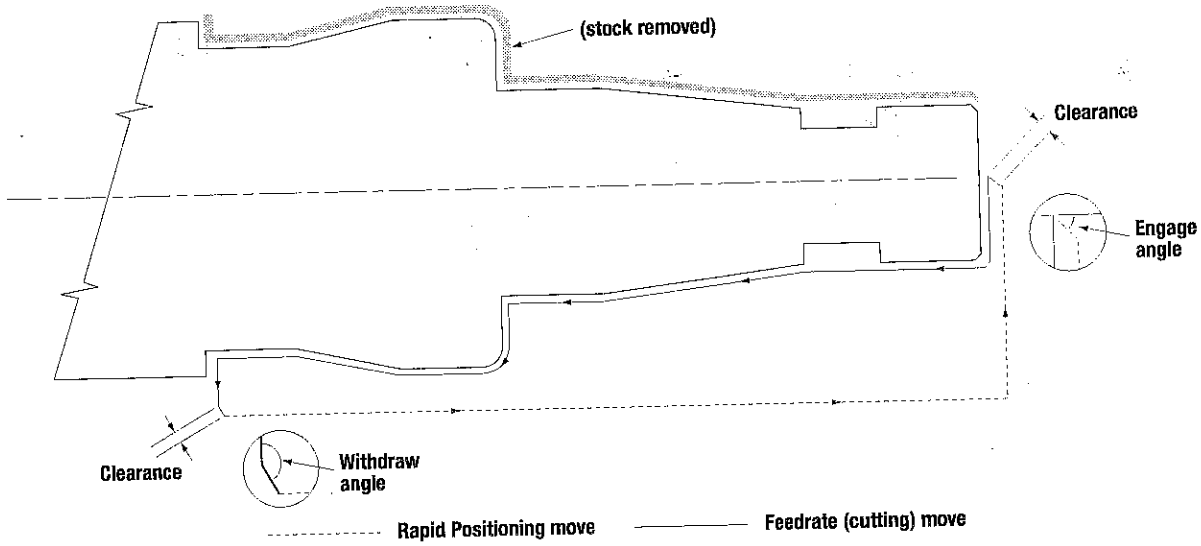
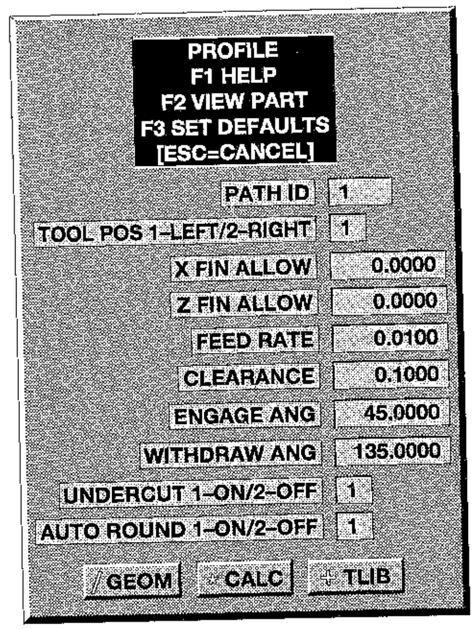


Figure 6-18 The tool motion caused by the Profile command. Profile only produces a single cutting pass.




```
0220 AUXFUN M8
0230 ROUGH 1 I1 X0.0100 Z0.0100 F0.0100 0.0100
  0.0100 S0.1000 C0.1250 W45.0000 W0.0500 D2
  U1 A1
```

6.3.3. Profile

F3 PROFIL

The Profile cycle basically causes the tool to travel the contour defined by the path from one end to the other. The tool motion is illustrated in Figure 6-18; the profile cycle here serves as a finishing operation to the roughing operation described in the previous section, removing the material that was left on the part as a finishing allowance. Profile produces a single cutting pass. When you select the command, a dialog box as shown in Figure 6-18 will prompt you for the following data:

PATH ID Enter here the name of a previously-defined path.

TOOL POS 1-LEFT/2-RIGHT Enter here where the tool will be in relation to the part, if you are looking in the direction of the tool motion. In other words, if the tool will be cutting from Z+ towards Z–, look in that direction, and input whether the tool will be to

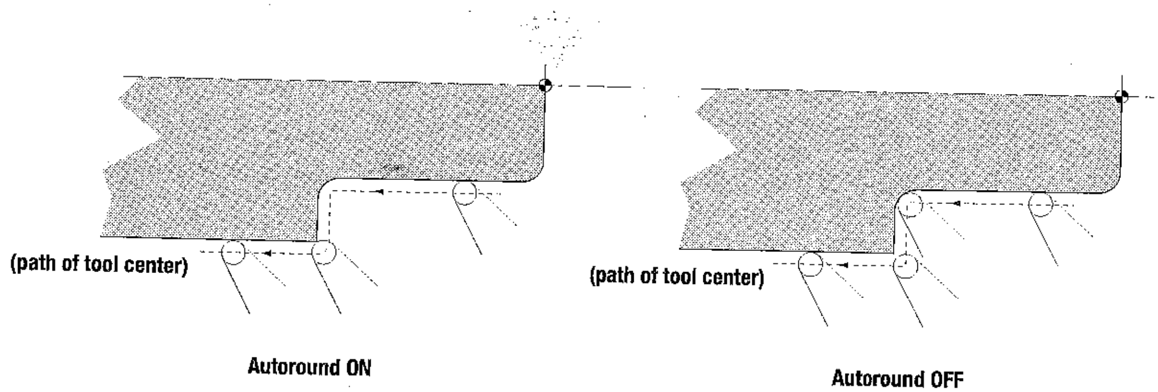


Figure 6-19 The autoround feature used by both Rough and Profile. Notice how when autoround is off, the tool gets pulled off the part when cutting a sharp corner.

the right of the part or the left.

X FIN ALLOW Enter here the amount of material that will be left on the diameter of the part, to be removed in a finishing operation. If the programmed diameter of the part is 1", and the finish allowance is .05", then the Profile cycle will only cut to diameter 1.05".

Z FIN ALLOW Enter here the amount of material that will be left on the vertical faces of the part, to be removed in a finishing operation.

FEED RATE This is the feedrate that will be used for any cutting moves.

CLEARANCE This is the distance from the part that the tool will move before and after the Profile cycle. This is a vector distance from the first and last points on the path: the clearance point at either end of the operation will be reached by moving away from the path at the engage or withdraw angle for the clearance distance.

ENGAGE ANG This is the angle at which the tool will approach the workpiece from the initial clearance point. Note that this is a cutting move at the feedrate.

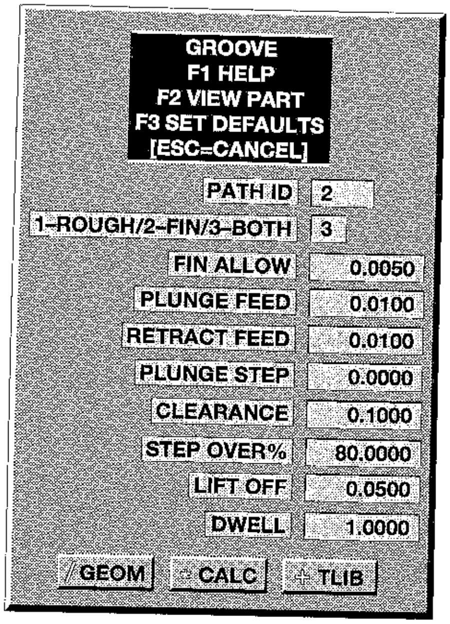
WITHDRAW ANG This is the angle at which the tool will retract from the workpiece at the end of the path. The angle is with respect to the direction of the last significant move, as illustrated in Figure 6-18. Note that this is *not* the same as the withdraw angle in Rough.

UNDERCUT 1–ON/2–OFF The EZPATH uses an undercut checking feature; when it is turned on, the EZPATH will check your program for undercuts when it loads. If one is found, it will tell you. Enter a 2 here to disable this feature.

AUTOROUND 1–ON/2–OFF The EZPATH also normally uses an autoround feature when calculating the path of the tool. This means that, when a sharp corner is programmed in the part, the path of the tool will have built into it a small arc at the corner equal to the radius of the tool, such that the tool will always remain in contact with the part and overall tool movement is minimized. The shape of the part is unaffected by this. If this is not desired, enter a 2 here. Figure 6-19 illustrates the effect.

The following program listing shows a path definition and the Profile instruction which calls it. This listing is similar to the sample program of section 6.3.1 (the last 4 lines are different). Note that Profile will cause the tool to move to the different points in the order in which they were entered in the path, whereas with Rough you could explicitly specify which direction to cut in. The data elements in the Profile instruction (line 0260) are in the same order as entered in the dialog box of Figure 6-18.

```
0010 STARTPATH 1
0020 RAPID ABS X0.0000 Z0.0000
0030 CHAMFER ABS X0.7500 Z0.0000 P0.0300 P0.0300 F0.0100
0040 LINE ABS X0.7500 Z-0.8750 F0.0100
0050 LINE ABS X1.0000 Z-1.8750 F0.0100
0060 LINE ABS X1.0000 Z-2.3750 F0.0100
0070 BLENDILN ABS X1.7200 Z-2.3750 R0.1000 CW F0.0100
0080 LINE ABS X1.7200 Z-2.8750 F0.0100
0090 LINE ABS X1.5000 Z-3.3750 F0.0100
0100 LINE ABS X1.5000 Z-3.8750 F0.0100
0110 LINE ABS X2.0000 Z-3.8750 F0.0100
0120 PATHSTOP
0130 STARTPATH 2
0140 RAPID ABS X0.7500 Z-0.5000
0150 LINE ABS X0.6500 Z-0.5000 F0.0100
0160 LINE ABS X0.6500 Z-0.8750 F0.0100
0170 LINE ABS X0.7500 Z-0.8750 F0.0100
0180 PATHSTOP
0190 RAPID ABS X2.0000 Z3.0000
0200 TLCHG I11 T01 01
0210 SETCSS G3 C675.00 S3200
0220 AUXFUN M8
0230 ROUGH 1 I1 X0.0100 Z0.0100 F0.0100 0.0100 0.0100 S0.1000 C0.1250W45.0000W0.0500D2U1A1
0240 RAPID  ABS  X2.0000  Z3.0000
0250 SETRPM  G3  S2120
0260 PROFIL  1  1  X0.0000  Z0.0000  F0.0100  C0.1000  E45.0000  W135.0000  U1  A1
```



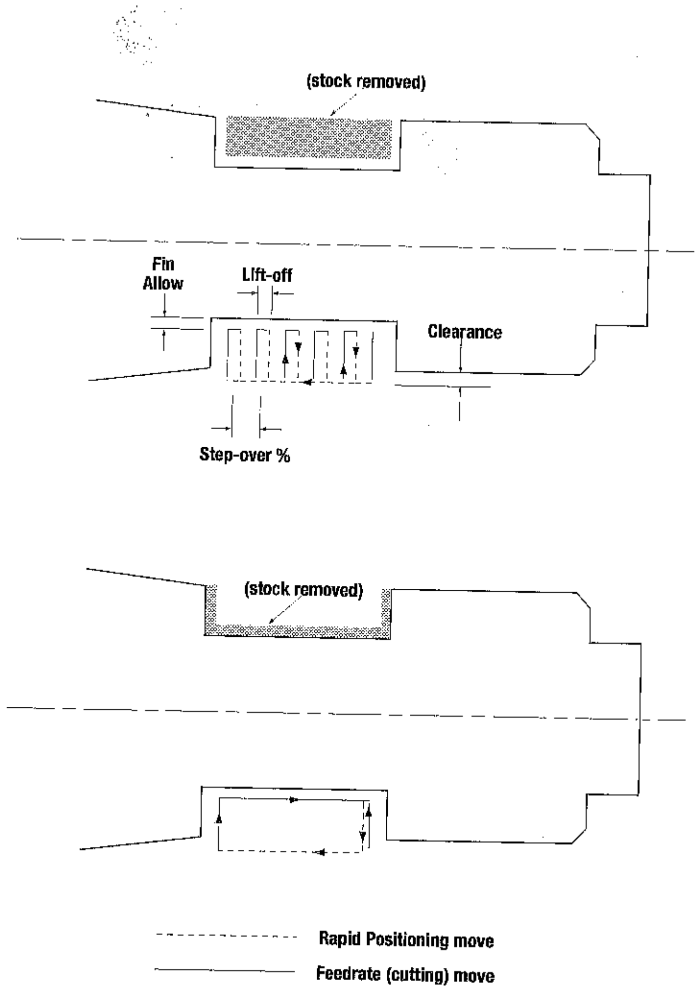


Figure 6-20 The Groove command. Notice the difference between the roughing and finishing passes.

6.3.4. Groove

F4 GROOVE

This cycle is specially designed to cut grooves in your part. You can integrate both roughing and finishing operations within a single command, or just perform either one. When you select the command, you will be prompted for the following information.

PATH ID Enter here the name of a previously-defined path.

1–ROUGH/2–FIN/3–BOTH You can use the Groove cycle for either roughing; finishing; or both. Enter 1, 2, or 3 as appropriate. Roughing and finishing are two distinct operations that each produce a different tool motion; if you select both, it will do first one, then the other. Figure 6-20 illustrates the different tool motions produced by each command.

FIN ALLOW Enter here the amount of material that will be left on the part after roughing—in each axis—to be removed by a finishing pass.

PLUNGE FEED This is the feedrate used when the tool is plunging into the part.

RETRACT FEED This is the feedrate that will be used during the lift-off move and any Z-axis motion during a finish pass.

PLUNGE STEP You can specify that the plunge move occur in a series of steps, whose value you specify here. If you just want a single step, leave this value at 0.

CLEARANCE This value determines how far from the part the tool is before beginning to feed into the part. It is measured from the highest point of the path. After each pass the tool withdraws to this plane.

STEP OVER % The step-over % determines how far the tool will advance in the Z-axis between each pass as a percentage of the tool width. For example, if the tool's width is .25", and the step-over % is 80%, the tool will advance .20" after each pass, so that each pass overlaps the previous one by .05".

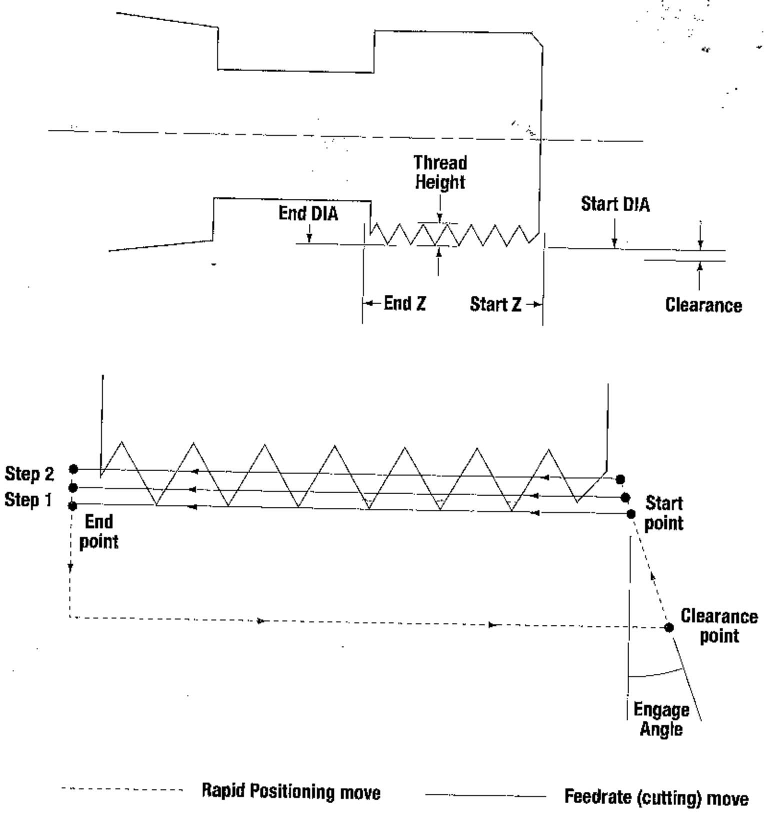
LIFT OFF After the tool has plunged to the programmed depth, it will move backwards by this amount—at the retract feed rate—before withdrawing at the rapid traverse rate. However, it does not do this after the first pass—it just withdraws to the clearance plane at the retract feed rate.

DWELL The tool can be set to remain in position at the bottom of each plunge move; specify a length of time here, in seconds, if you desire this.

Many of the above parameters apply just to the roughing part of the Groove motion. Figure 6-20 also illustrates the motion of the finishing pass, which resembles more the Profile cycle. Whatever material is left by the finishing allowance is removed in a single finishing pass, in which the tool moves along the contour of the path rather than in plunging steps. However, the motion along the path is modified so that the sides of the groove are cut only when the tool is feeding into the part. In Figure 6-20, the finishing pass finishes the first side of the groove; retracts and finishes the other side; then finishes the bottom surface.

The following program listing—excerpted from section 6.3.1—shows a path definition and the Groove instruction which calls it. The data elements in the Groove instruction (line 0270) are in the same order as entered in the dialog box of Figure 6-20.

```
0130 STARTPATH 2
0140 RAPID ABS X0.7500 Z-0.5000
0150 LINE ABS X0.6500 Z-0.5000 F0.0100
0160 LINE ABS X0.6500 Z-0.8750 F0.0100
0170 LINE ABS X0.7500 Z-0.8750 F0.0100
0180 PATHSTOP
  . . .
0270 GROOVE 2 3 A0.0030 F0.0100 R0.0100 P0.0000 C0.0130 O80.0000 L0.0500 D1.0000
```



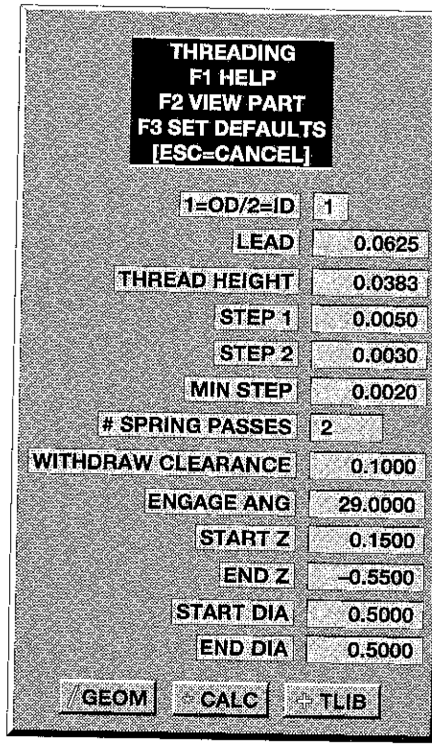


Figure 6-21 The Thread command. Once you enter a lead value, it will prompt you with a default thread height for a 60° thread.

6.3.5. Thread

F5 THREAD

Use the Thread cycle to precisely specify and control a threading operation. When you select the command, the dialog box shown in Figure 6-21 will ask you for the following data:

1=OD/2=ID Enter 1 or 2 here to indicate whether your threads will be cut on the outer or inner diameter of your part.

LEAD The lead distance is the distance the screw advances axially in one turn. This can be calculated as 1 divided by threads per inch. For example, if there are 13 threads per inch, the lead would be 1/13, or .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 will automatically be calculated for a 60° UN thread and inserted in this field; you can type over this value if you wish.

STEP 1 This is the cutting depth of the first pass of the threading tool.

STEP 2 This sets the cutting depth of the second pass of the threading tool. This will also be the depth of all the subsequent passes, until the amount of material left on the part equals the value of step 2. The EZPATH then reduces the cut depth by half for each step until the minimum step value is reached.

MIN STEP This determines the minimum step size that the EZPATH will use for this threading cycle.

SPRING PASSES Often, after a thread cycle is complete, it is necessary to make additional passes without changing the depth of the tool. Since the tool exerts a great deal of pressure on the part, the part might be slightly deformed while the threads are being cut, resulting in threads that are not cut accurately. Making additional spring passes with no change in the cutting depth ensures that the threads are cut as accurately as possible.

WITHDRAW CLEARANCE This is the distance from the part that the tool withdraws after each pass so that it can rapidly move back to the starting point to begin another pass. It is measured from the top of the threads.

ENGAGE ANGLE This is the angle at which the tool approaches the part from the clearance plane at the beginning of each pass. It is a positive angle, measured with respect to the X-axis.

START Z This is the Z-axis plane where the cutting begins. Usually this is close to but not actually on the part.

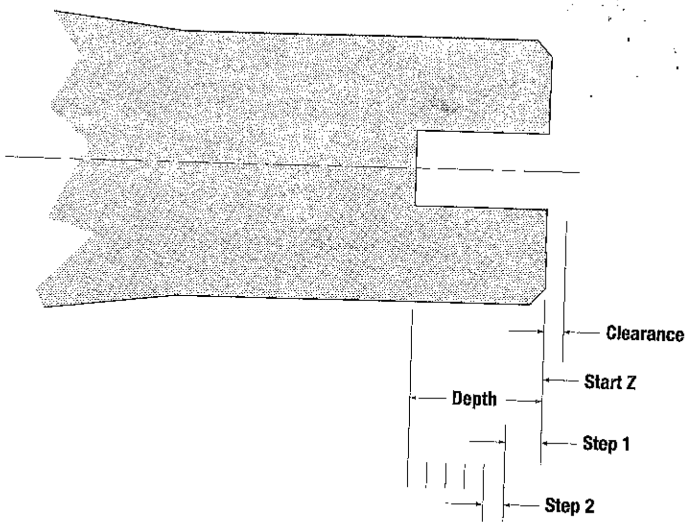
END Z This is the Z-axis plane where the cutting ends. Usually this is close to but not actually on the part. Generally, undercuts are made to each end where the threads will be cut. This is done before the threading operation so that the tool has sufficient room to clear the material before engaging or disengaging the part.

START DIA This is the diameter of the thread at the START Z position.

END DIA This is the diameter of the thread at the END Z position. This is usually the same as the START DIA; however, if the threads being cut are tapered threads or pipe threads, then this value is used by EZPATH to calculate the angle of the threads.

Figure 6-21 illustrates the geometry and tool motion of the Thread cycle. The data shown would produce the following command instruction:

0310 THREAD 1 L0.062500 H0.0383 S0.0050 0.0030 0.0020 #2 C0.1000 Z0.1500 -0.5500D0.7500 0.7500 29.0000



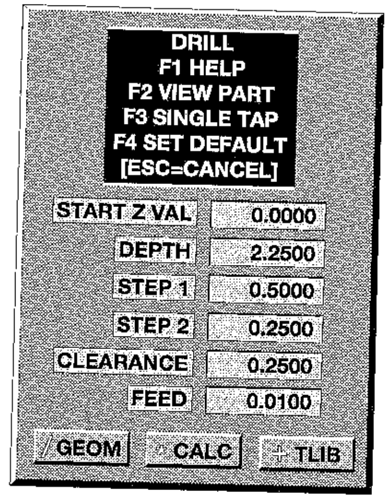


Figure 6-22 The part at left illustrates the Drill and Tap commands; with Tap, though, there are no pecking steps (**STEP 1** and (**STEP 2**) defined.

6.3.6. Drill

F6 DRILL

The Drill command is used to drill a hole in the center of the part. You can drill the hole in pecking depths, to clear chips and prevent overheating either the tool or part. The F6 DRILL key lets you access two different operations: a regular Drill cycle, described in this section, and a Tap cycle, which you can reach by hitting F3 when the Drill dialog box is displayed on the screen (Tap is described in the next section). When you select Drill, you will see the dialog box shown in Figure 6-22, prompting you for the following data:

START Z VAL Enter here the Z-axis position which defines the beginning of the hole. This is an absolute coordinate address, not an offset from the part surface.

DEPTH Enter here the total distance from the START Z VAL to the bottom of the hole.

STEP 1 This is the distance between the START Z VAL and the bottom of the first peck. To drill the hole in one pass, enter 0 here.

STEP 2 This is the distance of the second and each successive peck.

CLEARANCE This distance defines the clearance plane; it is measured as an offset amount from the START Z VAL. Before drilling begins, the tool will rapidly move to this point, and will withdraw here after each peck and after drilling is completed.

FEED RATE This is the feedrate for the drilling operation.

When your program reaches this command, the tool will rapid to the starting position, which is defined as X = 0 and Z = the start-Z value you just input plus the clearance distance. The

EZPATH will use the X=0 position as defined by the software (e.g. the active offset and/or work shift), regardless of the actual centerline of the spindle or tail stock.

While drilling, the tool movements are as follows:

1. After feeding to each peck depth, the tool will perform a rapid positioning move back to the clearance position. Note that the first pass will include both the **STEP 1** amount and the clearance distance.

2. It will then rapid back to the previously-drilled position minus the clearance distance.

3. It will then resume drilling at the feedrate through the clearance distance plus the step value.

4. After drilling is complete, the tool will rapidly move back to the original clearance plane.

Figure 6-22 illustrates the geometry of the Drill cycle. The data in the dialog box would produce the following instruction:

```
0090  DRILL  Z0.0000  D2.2500  T0.5000  0.2500  C0.1000  F0.0100
```

6.3.7. Tap

F6 DRILL : F3 TAP

After selecting Drill from the MDI screen, by pressing F3 you can program a single tap drill, where at the bottom of the hole the spindle will reverse and the tool will withdraw at the feedrate. Tap prompts you for the following data:

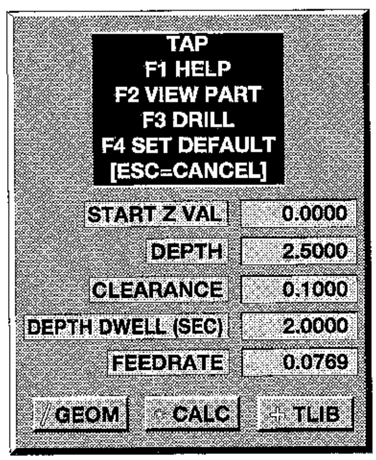


Figure 6-23 Data entry for a Tap cycle.

START Z VALUE Enter here the Z-axis position which defines the beginning of the hole. This is an absolute coordinate address, not an offset from the part surface.

DEPTH Enter here the total distance from the **START Z VAL** to the bottom of the hole.

CLEARANCE The point at which the tool will begin feeding into the part. This is measured as an offset from the **START Z** position. When the operation is finished, the tool will withdraw to this clearance point.

DEPTH DWELL (SEC) After it has drilled to the required depth, but before withdrawing, the tool will dwell (i.e. not move in X or Z while the spindle rotates) for the number of seconds you specify here.

FEEDRATE This is the feedrate used for both drilling and withdrawing.

When your program reaches this command, the tool will rapid to the starting position, which is defined as X = 0 and Z = the start-Z value you just input plus the clearance distance. Note that it moves to this position on a vector, instead of separate moves in each axis. It will then drill at the feedrate to the ending depth in a single pass. The spindle will reverse direction; the tool will remain in position for the dwell time; and then withdraw at the feedrate to the starting Z position. The spindle will reverse again, and then rapid to the clearance position.

The data shown in Figure 6-23 would produce the following instruction:

```
0010 TAP Z0.0000 D2.5000 C0.1000 T2.0000 F0.0769
```

6.4. Creating Programs with Save Point (Teach)

EZPATH provides a second, simpler way to create part programs: the Save Point (Teach) mode. After entering Save Point (Teach) mode, you can turn a part by using the handwheel or Jog mode to control the machine. Whenever you wish, you can tell EZPATH to remember the current tool position. It will then compile these points into a program that you can Edit or Run, just like any other program. You can enter Save Point (Teach) mode by pressing the [. SAVE PT] key from the Basic Operations screen. You can also enter Save Point (Teach) mode by pressing the [. SAVE PT] key while in Jog mode; for many users, this might be more convenient, because if you enter Save Point mode this way, you can use the [3 JOG X] key to toggle back to Jog mode.

After entering Save Point (Teach) mode, you will see the screen shown in Figure 6-24. The familiar large coordinate display shows the current position of the tool; the display area to the right of it shows the list of points that have been stored. You can exit and re-enter Save Point (Teach) mode without losing your current list of points; if you select a new tool or tool offset, EZPATH will automatically insert those commands in your Save Point (Teach) program.

The points you create are saved as EZPATH program instructions in a file called TEACH.PGM. Before the instruction for your first point, EZPATH automatically inserts a program instruction calling for a Tool Change to whatever tool and offset were effective when you entered Save Point (Teach) (see section 2.2.1 for more about Tool Change). After your last point, EZPATH inserts an Auxfun M2 instruction (section 6.2.10); this will Restart the program after each time you run it, so that you can easily run it several times in succession to repeatedly duplicate the part you've just programmed. You can use the EZPATH Editor to insert additional

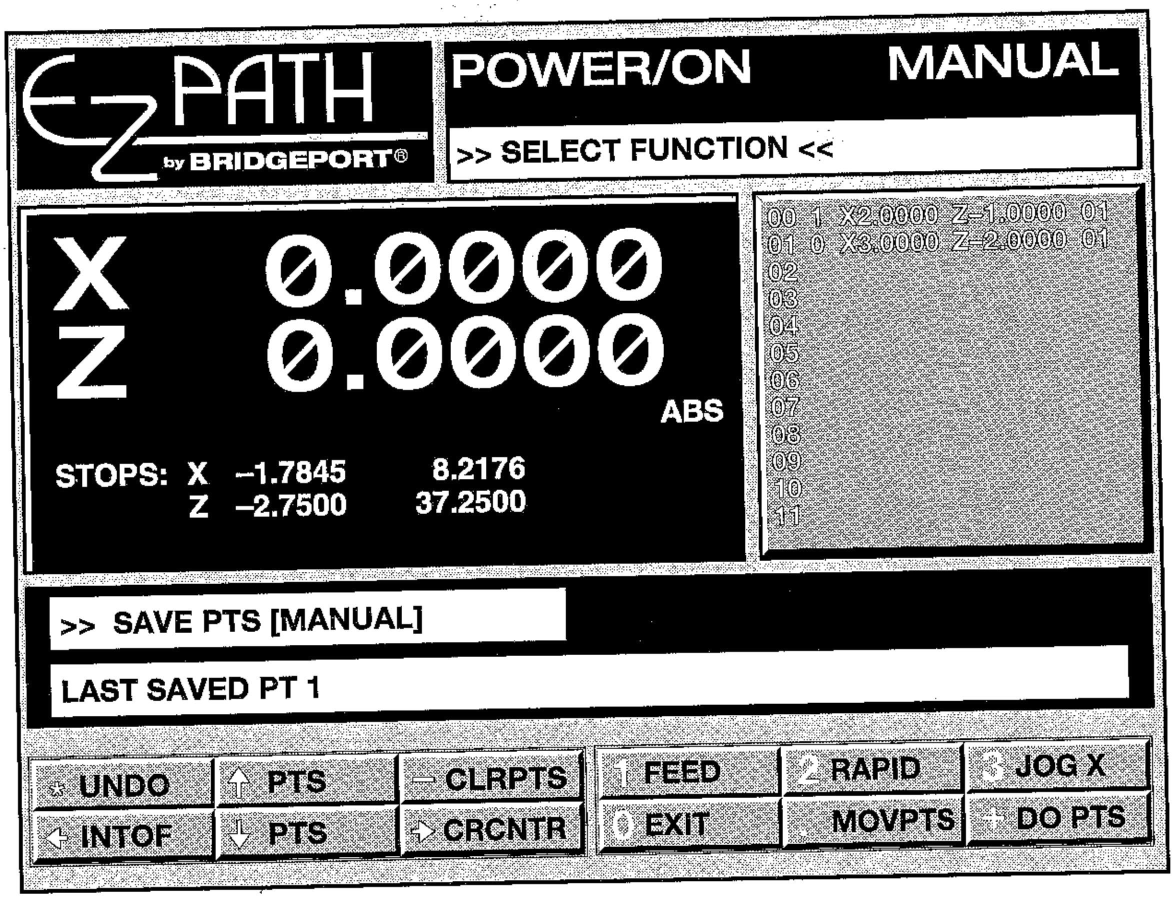


Figure 6-24 Save Point (Teach) screen, reached from the Basic Operations screen.

commands, rename the file, or make any other changes you wish (see Chapter 7 for a complete discussion of the EZPATH Editor).

All the points will be saved as absolute coordinates. While they are displayed in the window, you can scroll up and down the list by using the [↑ PTS] and [↓ PTS] keys.

The following commands are available in Save Point (Teach):

6.4.1. Feed to current point

[1 FEED]

After you have positioned the tool to the desired point, use this command to save the point as a feed event. In other words, EZPATH creates a command to cut on a line to this point. The following is an example of the program instruction it creates:

```
0010 LINE ABS X2.0000 Z2.0000 F0.3000
```

This is identical to a program instruction produced by the MDI Line command. The feedrate is specified when you exit Save Point (Teach) mode by pressing the [+ DO PTS] key; at that time, you can specify one feedrate that will be used for all of the Feed operations.

In the list of points on the right side of the display, points saved as Feed points have a 1 after the line number.

6.4.2. Rapid move to current point

[2 RAPID]

Use this command when you wish to execute a rapid—i.e. non-cutting—move to the saved point. The following is an example of the program instruction it creates:

```
0010 RAPID ABS X2.0000 Z2.0000
```

This is identical to a program instruction produced by the Position command in MDI mode (section 6.2.1). In the list of points on the right side of the display, points saved as Rapid points have a 0 after the line number.

6.4.3. Jog mode

[3 JOG X]

Use this key to switch to Jog mode so that you can precisely position the tool. You can use Jog mode as described in section 2.2.4. When you use the [0 EXIT] key to leave Jog mode, you will be returned to the Save Point (Teach) screen. Note that this key will only be available if you enter Save Point (Teach) from Jog mode.

6.4.4. Exit without Saving Points

[0 EXIT]

When you select this command, you will return to Basic Operations. Using this command will *not* save your points in the TEACH.PGM file on your hard disk; your list of points will be remembered, though, so that if you re-enter Save Point (Teach) you will see your data as you left it. Remember—you will not be able to edit or run your program unless you first save it to the EZPATH hard disk.

6.4.5. Move to a Point

[. MOVPTS]

Use this command to move the tool to a previously saved point. After selecting the command, you will be asked to input the number of a previously saved point. EZPATH will execute a rapid positioning move to this location, but will not save the move as a program instruction. Once the tool has been moved there, however, you can save that location using Feed or Rapid (as described above) just like any other point.

6.4.6. Do Points (Exit and Save)

[+ DO PTS]

When you select the Do Points command, EZPATH does a number of things:

- It will ask you to specify a feedrate that will be used by all of the feed commands in your Save Point (Teach) program.
- It will save the program created by your series of saved points in the program file called TEACH.PGM.
- It will then transfer you from Save Point (Teach) mode to Run mode.
- It will automatically load the program created from your series of saved points.

Once you are in Run mode, you can treat TEACH.PGM as you would any other program file. You can hit the [+ START] key to run it; you can View it (preview its execution on the screen); or, by hitting the [* EDIT] key, examine and edit it using the EZPATH Editor (see chapter 8 for a complete description of Run mode; see chapter 7 for the EZPATH Editor).

6.4.7. Intersection of two lines

[← INTOF]

Use this command to calculate the intersection of two lines. The two lines are specified by two points on each line; those four points must all be previously saved points, and they must all have been saved consecutively.

When you select the command, you will be asked for the number of just the first point. It then calculates the coordinates where the two lines intersect, and displays them for you.

You now have two options. You can press the [. MOVPTS] key to move to that point (exactly as if you were executing the Move to a Point command described in 6.4.5), or you can specify the number in the point list where it is to be stored. If there is already a point stored at that number, it will be overwritten without any warning.

6.4.8. Center of an Arc

→ CRCNTR

Use this command to calculate the centerpoint of an arc, based on the coordinates of three points on the arc. It works much the same as the Intersection command described in the previous section, in that the three points must be previously stored—consecutively—in your Save Point (Teach) list. When you select the command, you will be asked for the number of a point. EZPATH then looks at the three consecutive points beginning with the one you specified, and calculates the centerpoint coordinates—and radius—of the arc they share.

You have two options now—just like with the Intersection command. You can press the "." key to move to the centerpoint (exactly as if you were executing the Move to a Point command described in 6.4.5), or you can specify the number in the point list where the centerpoint is to be stored. If there is already a point stored at that number, it will be overwritten without any warning.

6.4.9. Undo points; Clear points

*UNDO ; - CLRPTS

Use these commands to delete points from your list. Pressing the *UNDO key deletes one point at a time from the list, beginning with the most recent; pressing - CLRPTS deletes all of the points from the list so you can start over.

Chapter 7
Editing Programs

The EZPATH II Editor is used to edit and modify already-created programs. You can use the Editor to edit programs that have already been saved on either the hard disk or a floppy diskette, or to edit a program that you are currently creating in MDI. To edit a program saved on disk, press the [5 EDIT] key on the Basic Operations screen; you can also enter the Editor from MDI ([+ EDIT] key) or Run mode ([* EDIT] key).

7.1. Using the Editor

7.1.1. Selecting a File

The first thing you need to do after entering Edit mode is choose a program to work on. When the Editor is selected from the Basic Operations screen, you will see a window like Figure 7-1; it shows you all of the program files on the hard disk. Use the ↑ and ↓ arrow keys to position the highlight bar over the program you wish to edit, and press [+SELECT]; you will see the program text in a new window. To edit files on the floppy drive (A:\), press F5.

After a program has been loaded, you will see the screen shown in Figure 7-2.

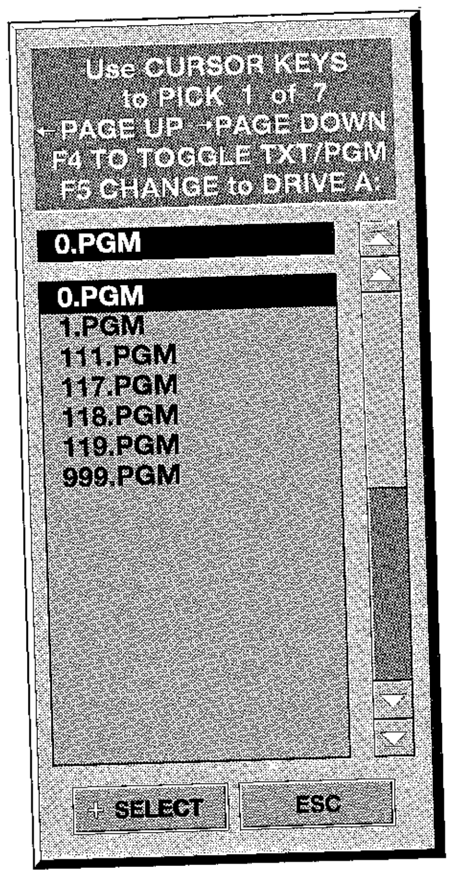


Figure 7-1 Selecting a program file to edit; hit F5 to edit a file stored on a floppy diskette.

7.1.2. Cursor movement keys

When the program file is displayed, use the cursor movement keys to select lines to edit. [↑ UP] and [↓ DWN] move the cursor in that direction one line at a time; [← TOF] moves to the first line in the program; [→ EOF] moves to the last line. The number of the current line (the *sequence number*) is displayed in the upper left corner of the Edit window.

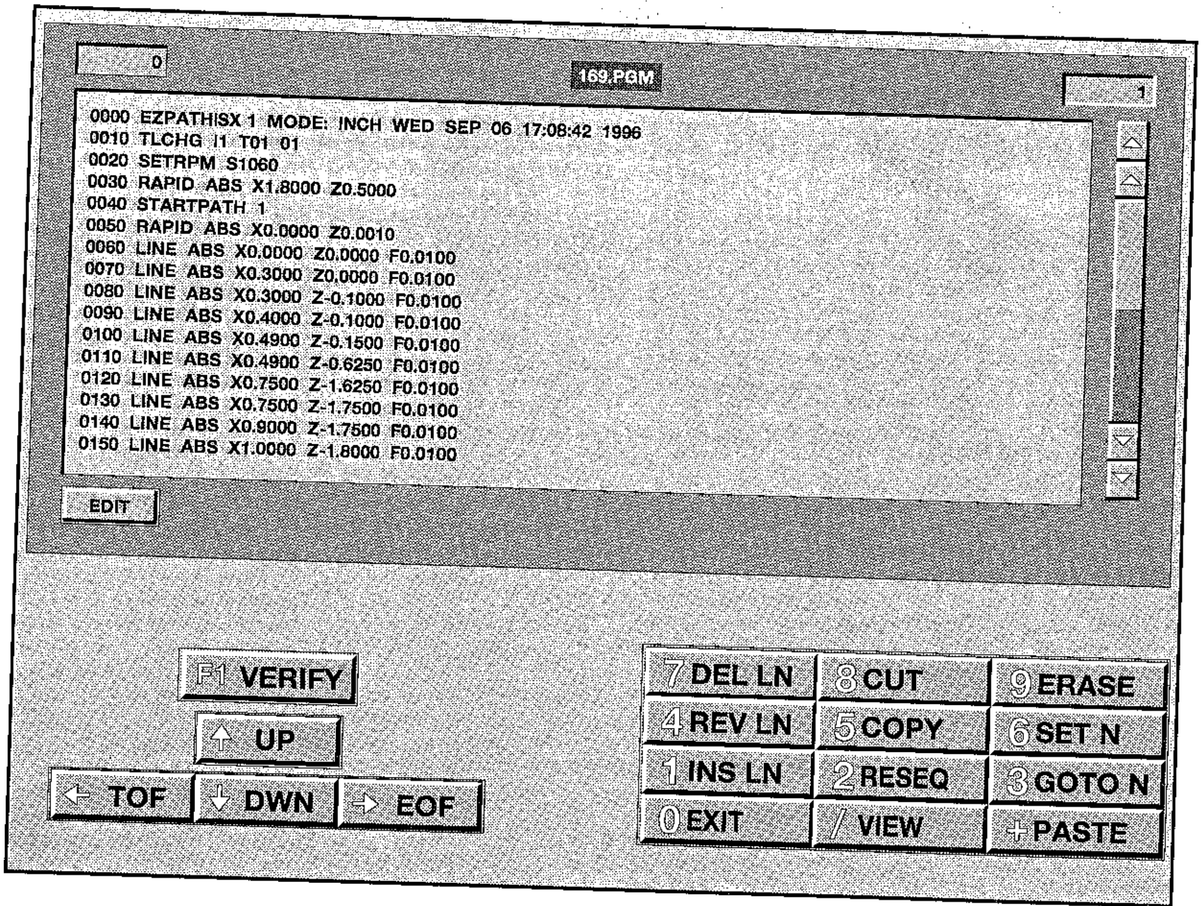


Figure 7-2 The Editor command screen. First, use the arrow keys to select a line to edit; then, select a command.

7.2. Editor commands

7.2.1. Insert Line

[1 INS LN]

Once you have chosen a program to edit, this command lets you insert a new line (or as many lines as you wish). The new lines will be inserted *after* the current line. When you select the Insert Line command, EZPATH places you in MDI mode; you can now select commands in exactly the same manner as you normally would in MDI (described in Chapter 6). You can return to the Editor by pressing [+ EDIT] or [0 EXIT].

Although the new lines will be inserted immediately after the current cursor point, they will be numbered in accordance with the settings specified with the Set Number command; see section 7.2.6 for details.

If you want to enter new lines after the end of your program, before pressing the [1 INS LN] key be sure that the cursor is on the last line—not on the blank line after it.

7.2.2. Resequence

2 RESEQ

This command renumbers all of the lines in the program. After you have inserted new lines, or moved or deleted lines, the sequence numbers might no longer be in numerical order. Use this command to correct this. When you select this command, it will ask for the number you wish to use for the first program line (N,SEQNO), and the increment for each successive line (N,INC). It will automatically pad the line numbers with zeros so that each line number is 4 digits long. For example, entering 0 and 10 in response to the prompts will result in line numbers of 0000, 0010, 0020, 0030, etc.

If, at some point after Resequencing, you wish to Insert new lines using 1 INS LN—and have not set line numbering with Set N—they will be numbered according to the settings just used in Resequencing.

7.2.3. Go To line number N

3 GOTO N

Use this command to jump to a specific line number. Go To only works from the current cursor position forward to the end of the program; it will not search backwards. If the line number you enter does not exist, it will move you to the end of the program. You do not have to enter leading zeros; to move to line 150, for instance, enter 150, not 0150.

7.2.4. Review Line

4 REV LN

As you examine the lines of program instructions, some of the information might seem a little bit cryptic. If you position the cursor on a line and press the 4 REV LN key, you will see the original dialog box used to create the instruction, with the name of the command and all of the original data elements.

Review Line will also allow you to change the data or parameters used by that line's command; just enter new data in the dialog box. It will not, however, allow you change the command itself. For example, if the line contains a Tool Change command, you can use Review Line to change the offset number or tool ID, but you cannot use it to replace Tool Change with an entirely different command. To do that, you must delete the line, and use Insert to place the new command.

Note also that the first line in the part program (*e.g.* 0000 EZPATH 1 MODE I INCH) cannot be edited.

7.2.5. Copy; Cut; Paste

5 COPY ; 8 CUT ; + PASTE

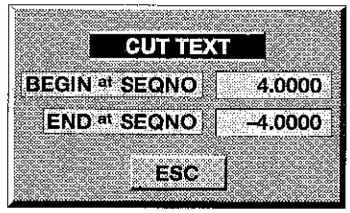


Figure 7-3 Selecting a series of lines to be cut. Selecting lines to be copied is identical.

These commands let you copy or move a block of lines within your program. To *copy* a line or group of lines, select Copy; to *move* a line or block of lines, select Cut. EZPATH will ask you for a range of line numbers: where it says **BEGIN**, enter the number of the first line to be copied or moved; where it says **END**, enter the line number *following* the last line to be included. For example, to copy lines 80–110, enter 80 in the **BEGIN** box, and 120 in the **END** box. Now move the cursor to wherever you wish the lines to be, and press + PASTE; it will insert the text after the cursor.

Please note the following considerations when using these commands:

1. Before beginning the Copy or Cut operation, the cursor must be at a point in the program before the first line in the selected block.

2. To select a single line, the **BEGIN** line number should be that line, and the **END** line number should be the number of the next line. For example, to move line 0090, the **BEGIN** line should be 0090, and the **END** line should be 0100.

3. To move or copy lines after the last line in the program, before selecting Paste the cursor should be positioned on the last program line. If it is positioned anywhere after the last line, Paste will not work.

4. When you Cut or Copy, the selected data is actually copied to a disk file called COPY.TMP, and is available for pasting until you Cut or Copy new data. You can paste the data multiple times in different places; you can even save the current file, open a new one, and paste the data into the new file.

5. The line numbers are not renumbered after this operation. If you paste the same data multiple times, you will have lines with duplicate line numbers. Remember to use Resequence to renumber your program lines correctly.

7.2.6. Set Number

6 SET N

Use this command to set the line numbers of any new lines which will be inserted into the program. **N,SEQNO** will be used as the number of the first inserted line; each successive line will be incremented by **N,INC**. If you Insert lines more than once, EZPATH will keep track of the line numbers so that it will not keep beginning new lines with the same number.

7.2.7. Delete Line

7 DEL LN

This command deletes the current line. It will not delete the first line in the program.

7.2.8. Erase

9 ERASE

Use Erase to delete a block of lines at one time. Erase will ask you to specify a block of lines in exactly the same way as Copy and Cut; see 7.2.5 for details. The difference between Cut and Erase is that Cut will copy the data to a buffer file, so that you can retrieve it with Paste (until you Cut something new); with Erase, the data is permanently gone as soon as you execute the command.

7.2.9. Exit Editor

0 EXIT

Use this command to exit the Editor and save the edited program. When you select this command, a new window pops up which will give you several different options.

In the space next to **PRGM**, you can enter a new name for your program. You can use any of the number keys to create a new name—up to a maximum of 8 characters. The file extension .PGM is added automatically. If you do not enter a new name (or enter the name of a program which already exists) the existing program with that name will be overwritten.

You can now save your program in one of three ways using the following keys:

+ SAVE saves the program under the name listed above in the **PRGM** field, and then exits.

* SAVE|RUN saves the program and then loads it into memory so that it can be run (see section 8.1 for information on Run mode).

/ SAVE|VIEW saves the program and then lets you view it in Run Preview mode (see section 8.2 for information on Run Preview).

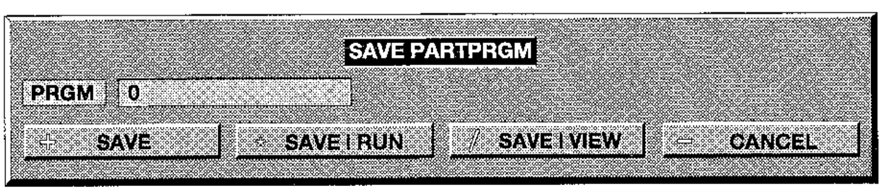


Figure 7-4 Exiting the EZPATH Editor. Whenever you try to exit the Editor, it will prompt you to save your file, and any changes you have made.

ESC exits the Editor without saving the part program. Any changes that were made to the program are not saved.

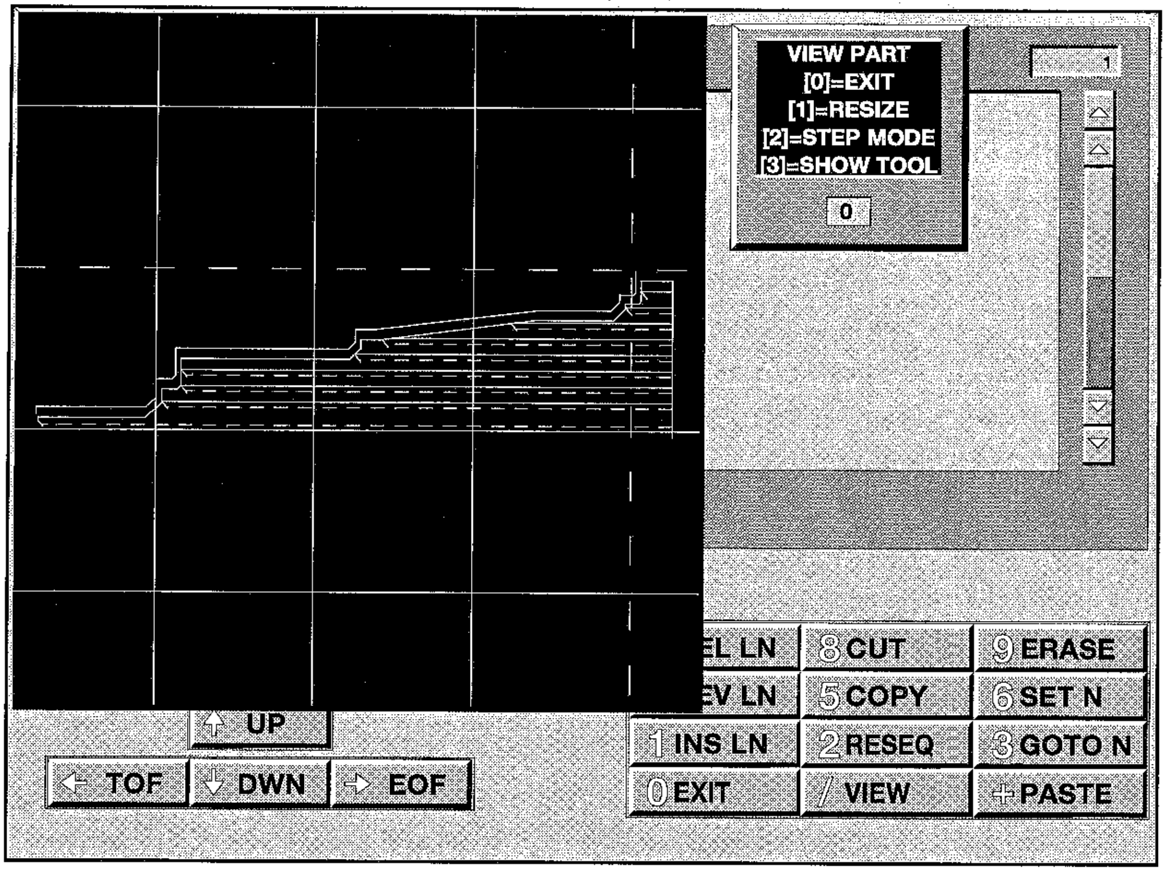


Figure 7-5 The Verify display. While View shows you the shape of the part created by your program, Verify lets you focus on a particular canned cycle operation, studying the tool motion as well as the part shape.

[- CANCEL] cancels the Exit command and returns to the Editor without saving the part program.

7.2.10. View

[/ VIEW]

The View command displays the part program geometry on the screen, for either the whole program or a range of instructions; you will be prompted for the first and last line numbers to be viewed. After the part is viewed, you have the option to either Exit back to the Editor, or Review the part.

7.2.11. Verify

[* VERIFY]

This command is used to check the actual movement of the tool during a canned cycle operation—either Rough, Profile, Groove, Thread, or Drill/Tap (see section 6.3). If you try to Verify any other operation, you will get a message that ONLY CANNED CYCLES CAN BE VERIFIED. After positioning the cursor on the line which contains the appropriate command and pressing the [* VERIFY] key, you can set several viewing options:

0 EXIT returns you to the Editor.

1 RESIZE allows you to zoom in on a particular segment of the part and view the tool motion in great detail; it asks you to specify the maximum and minimum X- and Z-axis positions that will be covered by the viewing window.

2 STEP MODE lets you view the tool motion one step at a time or all steps in rapid succession. After pressing 2, press 0 for single-step viewing, or 1 for continuous. In single-step mode, press [ENTER] after each step. At any time, you can press 1 to change to continuous-view mode.

3 SHOW TOOL displays the tool tip while the preview is being shown. If you press the 3 key, and then select step mode, you can go through step mode showing the tool path.

Note the difference between Verify and the View command described in the previous section. The View command can only show you the part; the Verify command lets you see the tool motions, step-by-step if you choose, in addition to the part.

Chapter 8
Running Programs

EZPATH II's Run mode is used to execute a program. You can run a program either continuously or in single-step mode (section 8.1), or you can use the View feature to preview your program's operation on the screen before cutting an actual part (section 8.2).

To run an existing program, enter Run mode by pressing the [+ RUN] key from the Basic Operations keyboard. You can also run or preview a program after editing it with the EZPATH Editor, or after creating and saving it in MDI mode.

8.1. Run mode

When you enter Run mode, you will see the screen shown in Figure 8-2, illustrating the commands available to you.

While the program is running, the line labeled ACTV in the middle of the screen displays the program block which is currently being executed; the line above it (PREV) displays the previous block, and the NEXT line displays the next block to be executed. Note that the instructions displayed here will be G-code blocks, even if you've created the program in MDI.

8.1.1. Loading a program

[← LOAD]

Before a program can be run, it has to be loaded into memory. This command lets you load a program that has been saved on either the EZPATH hard disk or a floppy diskette. After selecting this command, you will see the box shown in Figure 8-1. Use the arrow keys to highlight the program file you wish to run; the F4 key to view either TXT or PGM files; and the F5 key to toggle between the floppy diskette drive (A:\) and the hard disk (C:\). When you've highlighted the one you want, load it by pressing the [+ LOAD] key.

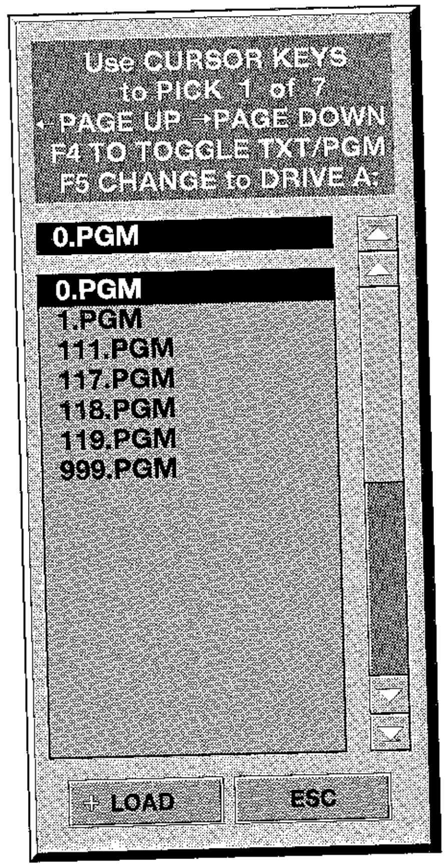


Figure 8-1 Before a program can be Run, it has to be loaded into memory.

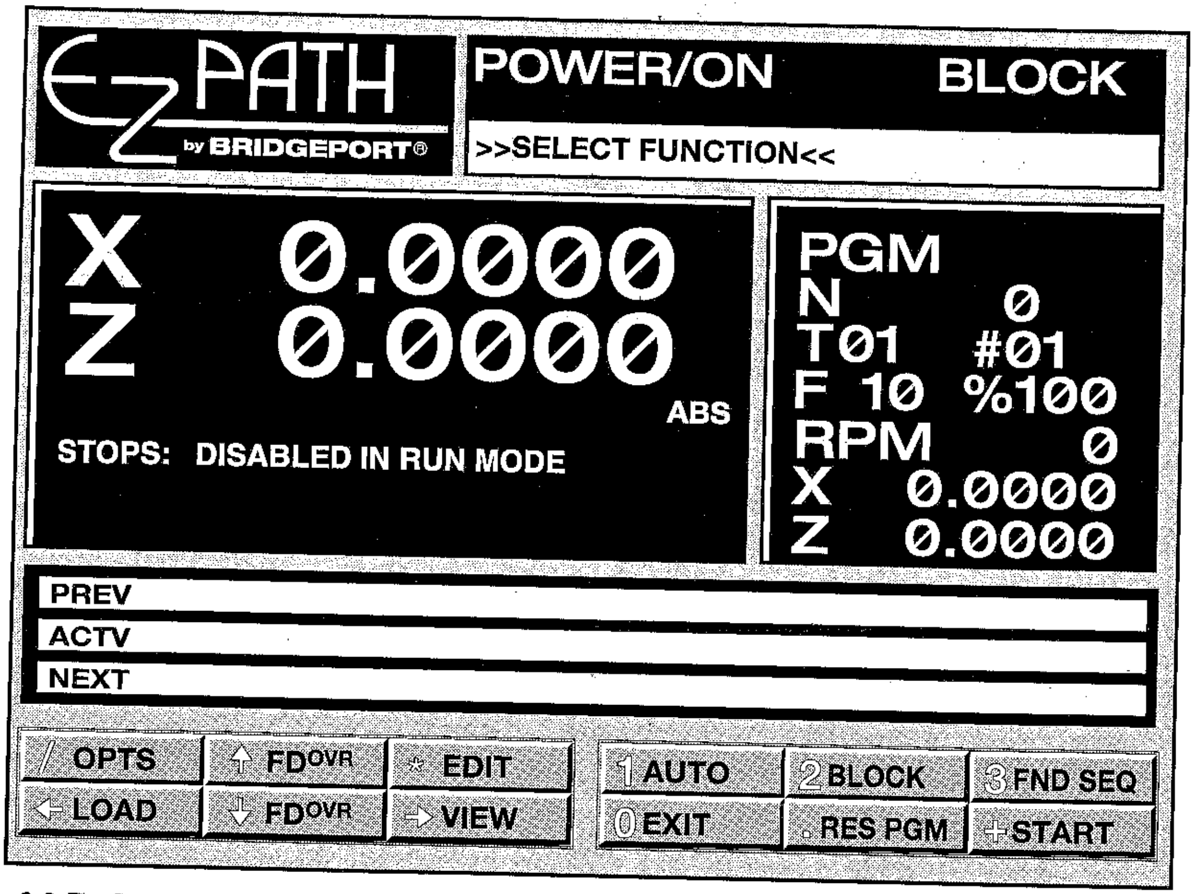


Figure 8-2 The Run mode screen lets you control the execution of a program. While a program is running, the last instruction to be executed, the current instruction, and the next instruction are always displayed for your information.

8.1.2. Automatic/Block execution

[1 AUTO], [2 BLOCK]

You can set the EZPATH to run your program continuously; or you can have it execute 1 line at a time. If you run in Automatic mode, the program will stop for tool changes; programmed stops; or when the Hold button is pressed. Program execution begins when you press the [+ START] key or the Run button. The status indicator in the upper-right corner of the screen will display AUTO.

If you select Block mode, a block of program instructions will be executed each time you press the [+ START] key. At any time, you can execute the rest of your program in continuous mode by pressing the "1" key. While you are in block mode, the status indicator in the upper-right corner of the screen will display BLOCK. Note that "block" here refers to blocks of G-code instructions, not MDI program lines. Some MDI commands—notably the canned cycles—can each produce several G-code blocks.

8.1.3. Find Sequence number

3 FND SEQ

This command allows you to find a specific line in your program and begin program execution with that line. After selecting the command, it will ask you for a SEEK NUM; enter the appropriate sequence number (line number) of your program. If you press the - T,TOOL key, you will be asked instead for the number of a tool; EZPATH will then search your program for the instructions specifying that tool, and let you begin program execution there. Press the + N,SEQ key to search again for a sequence number.

Please note that this command only searches down through the program, so that if the line you are searching for comes before the current line, EZPATH will be unable to find it. It may be necessary to Resequence the program (see section 7.2.2) or Reset it (next section) before using this command.

8.1.4. Reset Program

. RES PGM

Use this command to Reset the loaded program, making the first line in the program the current line. After a program is run, it must be Reset before it can be run again.

8.1.5. Start Program

+ START

Use this command to begin execution of a program, or to resume execution if it has been stopped or paused.

8.1.6. Run Options

/ OPTS

By selecting this command you can enable/disable the Optional Stop and Block Delete program instructions (see chapter 6.2.10). When creating a program in MDI, you can insert optional stop commands; the program will stop at these points only if Optional Stop is enabled here in Run mode.

In MDI, you can also insert Block Delete Start and Block Delete End commands; these mark the beginning and end of sections of a program that will only be executed if Block Delete is disabled. In other words, if a section of the program is marked for Block Delete, those commands will *not* be executed if Block Delete is turned on.

You will use these options most often when testing new programs.

The Run Options dialog box also lets you turn the coolant on and off. Unlike the other Run

Options, this feature is not linked to anything in MDI or your program; it works just like a hard-wired switch. The coolant *is* linked to the spindle power, though, so that the spindle must be on before this command can work.

8.1.7. Feed Override keys

[↓ FD OVR], [↑ FD OVR]

Use these keys to set the actual feedrate as a percentage of whatever is programmed. You can specify that the actual feedrate be anywhere from 0–150% of the programmed value. Each key increases or decreases the feedrate by 5%; you can see the effect in the status window (see section 1.2.4).

8.1.8. Edit Program

[* EDIT]

This command calls up the EZPATH Editor, which you can use to view program instructions or make changes to the current program. See Chapter 7 for a detailed description of the EZPATH Editor.

8.1.9. Exit

[0 EXIT]

Select this command to Exit Run mode. EZPATH will not let you exit while a program is running; stop program execution with the Stop button, if necessary, before exiting.

After you exit Run mode, your program remains in memory (at least until another program is loaded). However, if you wish to run the program again, you must use the Reset command.

8.2. View mode

While in Run mode, you can use the View command ([→ VIEW] key) to preview program execution on the screen before actually cutting the part. The screen displays a 2-dimensional view in the XZ plane; you can also view the tool in addition to the part. Note that the View command described here is different from the View command in the Editor (section 7.2.10): the View command in the Editor only shows you the contour of the part, while View command here also shows the tool motion, simulating the execution of the program.

After selecting the command, you will see the screen shown in Figure 8-3. The large box is the viewing window; in the upper right is a representation of the EZPATH status display. It displays (from top to bottom): the name of the running program; the current line number; the tool number; the feedrate; and the current X- and Z-axis positions.

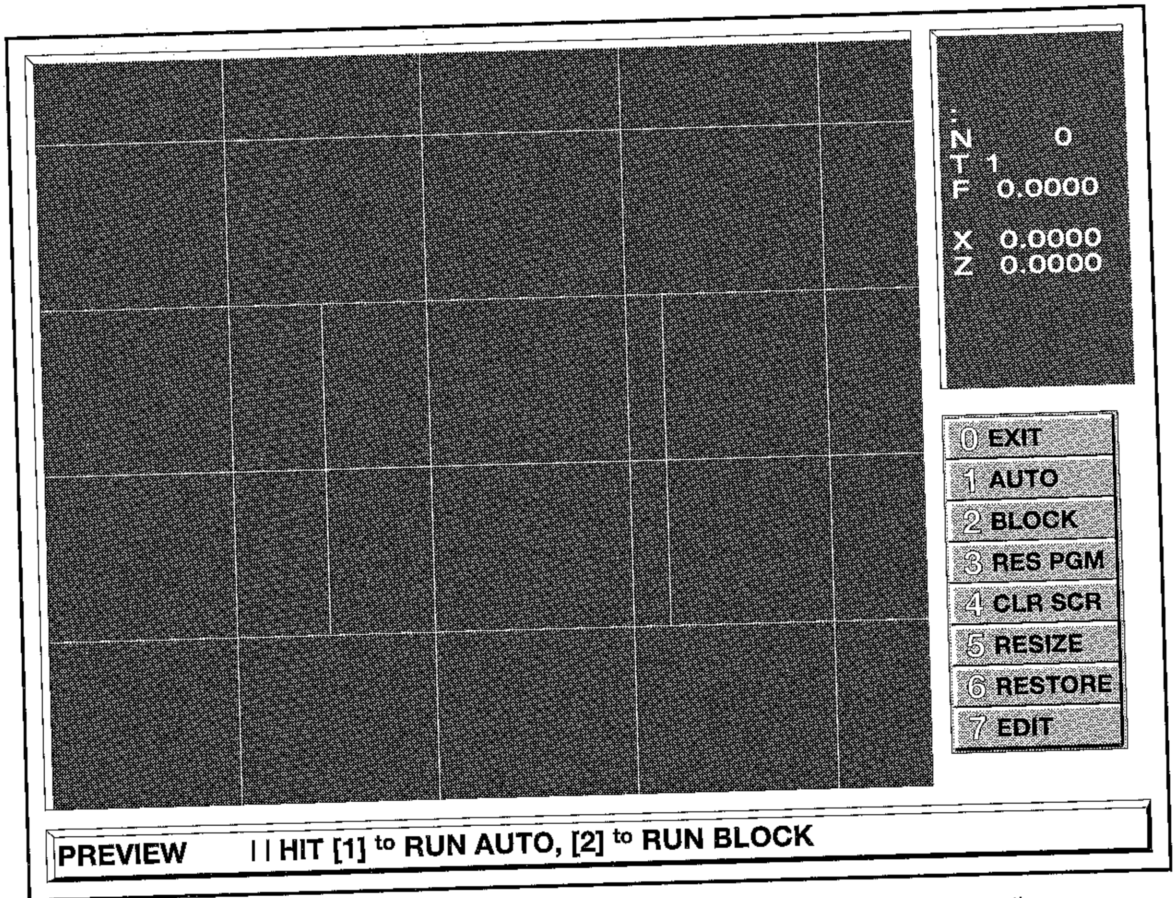


Figure 8-3 The Run: View screen. Use the View command to preview the program's execution on the screen before cutting a part; the options on the right side of the screen let you control the program's execution the same way you would if you were running it for real.

While the program is running, cutting moves will be shown on the screen as solid lines; rapid moves will appear as dotted lines. Program execution will stop for programmed stop events (like a tool change) just as if you were cutting a part; press the [+START] key to resume execution. While in View mode, you can use the following commands:

8.2.1. Exit

[0 EXIT]

After using this command, you will be returned to Run mode. Please note that if you now want to run your program for real, you must Reset it (see chapter 8.1.4 above).

8.2.2. Automatic/Block execution

[1 AUTO], [2 BLOCK]

These commands function the same as their Run-mode namesakes (chapters 8.1.2 above), except that after pressing one of these keys the program preview will begin immediately; in other words, after selecting one of these commands, you do not have to press [+ START]. While in block mode, you always have the option of executing either the next block (by hitting "2"), or the remainder of the program in continuous mode (by hitting "1").

8.2.3. Reset Program

[3 RES PGM]

Use this command after you have previewed all or part of a program and wish to View it again from the beginning. It is exactly the same as the Reset Program command in Run mode (see chapter 8.1.4 above). You might wish to Clear the Screen (next section) before viewing your program again.

8.2.4. Clear screen

[4 CLR SCN]

Use this command to clear the viewing screen so that data from old previews does not clutter up your current display.

If your program is paused for a tool change, you might wish to use this command to clear the tool path of the previous tool, so you can see more clearly the motion of the current tool. Just follow these steps:

1) While the program is paused, hit the "0" key to stop program execution;

2) Hit the [4 CLR SCN] key to clear the screen;

3) Hit the " + " key to resume program execution.

8.2.5. Resize

[5 RESIZE]

This command lets you zoom in on a small section of the part so that you can view the program execution in greater detail. It will prompt you for X- and Z-coordinates which will become the boundaries of the viewing window. X MIN sets the bottom of the viewing window; Z MIN the right edge; X MAX the top; and Z MAX the left edge. (After you enter all of the coordinates, EZPATH might alter them slightly to completely fill the viewing window.)

The next three parameters control how the tool and motion will be displayed. SHOW TOOL will cause the tip of the tool to be displayed while the program is previewing (if you enter 1 in the box). If you enter 1 in the SHOW PATH box, the path of the tool will also be displayed. The third box (TRACKING PT) lets you specify whether the displayed path will be based on the touch-off point or the actual center of the tool.

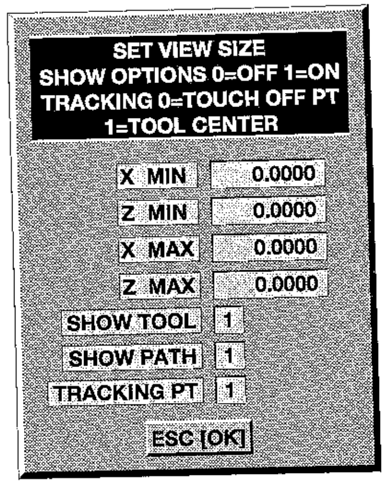


Figure 8-4 Use Resize to scale the size of the preview area and also to control how the tool is displayed.

8.2.6. Restore

6 RESTORE

Restore returns the viewing window to its original settings, if you have changed them with Resize.

8.2.7. Edit

7 EDIT

This command calls up the EZPATH Editor, which you can use to view program instructions or make changes to the current program.

Chapter 9
File Utilities

The EZPATH II provides you with a complete set of file utilities which allow you to copy data to and from the EZPATH's hard disk; floppy diskettes; or directly from another computer. You can access these utilities by selecting 9 UTILS from the Basic Operations screen. The familiar EZPATH screen will disappear, replaced by the display shown in Figure 9-1. The commands are described in the following sections.

9.1. <1> Complete Diskcopy

Use this command to copy one entire floppy diskette onto another. This will require swapping the diskettes at least three times. If you are copying the EZPATH system diskette, make sure the system

```
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:
```

Figure 9-1 Utilities menu

```
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:
```

Figure 9-2 Utilities: Copy files (option <2> from main menu). Use these commands to copy files from the EZPATH hard disk to a floppy diskette.

diskette is locked before copying; this will prevent it from being accidentally overwritten. To lock a diskette, slide the small tab on the back of the diskette so that the hole is uncovered. If you need to use this diskette to restart the EZPATH, remember to slide the tab back so that the hole is covered up.

9.2. <2>, <3> COPY files

These two menu options allow you to copy files back and forth between the EZPATH hard disk and a floppy diskette. Program files and related data are usually stored on the machine's hard disk so that they are easily available; on the other hand, a permanent copy is usually stored on a floppy diskette to be used as a backup copy. Option 2 lets you copy files from the EZPATH hard disk to a floppy disk; use option 3 to copy files in the other direction (from a floppy onto your hard disk). A 3.5-inch diskette must be in the disk drive before selecting either command.

When Option 2 is selected, the following choices are shown on the screen (see Figure 9-2):

```
<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:
```

Figure 9-3 Utilities: Copy files (option <3> from main menu). Use these commands to copy files from the floppy diskette to the EZPATH hard disk.

Selecting 1 will display a list of all the EZPATH files on the hard disk; selecting 2 displays a list of all such files on the floppy diskette. (A EZPATH file is a program file that has been created directly on the EZPATH machine; its filename will end with .PGM.)

Selecting 3 will display a list of OTHER files on the hard disk; these will be program files that have *not* been created on the EZPATH machine, and whose filenames will end in .TXT. Selecting 4 lists all of these files on the floppy diskette.

Selecting 5 copies *all* of the PGM files from the hard disk to the floppy; selecting 6 copies all of the TXT files.

Pressing the 7 key lets you choose one PGM file, by name, and copy it to the floppy disk; 8 lets you copy one TXT file in the same way.

Option 9 lets you select a tool library and copy it to the floppy disk.

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

All of the above commands copy data from the hard disk to a floppy diskette; if you want to move files from a floppy diskette to the EZPATH hard disk, select 3 from the Utilities main menu; you will see the screen shown in Figure 9-3.

As you can see, it is identical to the screen shown in Figure 9-2, except that options 5–9 copy data in the other direction: from the floppy diskette to the hard disk. Use these commands as described above.

9.3. <4> DELETE files

Option 4 from the Utilities main menu is used to delete files from either the floppy diskette or the hard disk. Selecting it displays the options shown in Figure 9-4.

Options 1–4 work the same way as in the Copy Files menus; they display lists of PGM and TXT files from the hard disk and floppy diskette.

Select option 5 to delete one or more PGM files from the EZPATH hard disk. After you press the 5

```
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:
```

Figure 9-4 Utilities: Delete Files menu (option <4> from main menu).

key, it will ask you to specify—in either of two ways—the file(s) you wish to delete: you can either type the file name and it will delete that one file; or you can type *, and it will delete *all* .PGM files on the hard disk! *Be extremely careful about deleting files using * !!* If you hit ENTER without typing anything, you will return (harmlessly) to the Delete menu.

Options 6, 7, and 8 work in the exact same manner, allowing you to delete TXT files from the EZPATH hard disk (6); PGM files from a floppy diskette (7); and TXT files from a diskette (8).

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

9.4. <5>, <6> VIEWING a file

Use these utilities to examine the contents of a file on the screen. Selecting this option will only display the file; you cannot edit or make any changes to the file. Use 5 to view files on a floppy diskette, and 6 for files on the hard disk.

When 5 is selected, the following choices are shown on the screen (option 6 would be identical, except that "floppy disk" would be "EZPATH").

Pressing the 1 key lists all the PGM files that are on the floppy disk in the disk drive. If you press the 2 key, it will let you select a PGM file to display. Options 3 and 4 work the same way for TXT files.

As always, pressing the ESC key exits this menu, and returns to the main Utilities 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:
```

Figure 9-5 Utilities: View Files menu (option <5> from Utilities main menu). Use this command to view files on the floppy diskette; use main menu option <6> to view files on the hard disk.

```
SEND-RECEIVE Protocol

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

<ESC> QUIT.

>>> Select:
```

Figure 9-6 Selecting a communications protocol

9.5. <7> SEND or RECEIVE files

EZPATH can also share data and download programs from other computers (such as a computer running EZ-CAM software) or another EZPATH machine. Data is transferred via a cable connected to the machines' serial ports (also known as RS-232 ports). Bridgeport Machines EZ-CAM adapter cable (part 1940515) and Universal Communications cable (part 1940303) can be used for this purpose.

9.5.1. Communication Protocols

After selecting 7 from the Utilities main menu, you will see the menu shown in Figure 9-6. Before the two machines can communicate with each other, you need to choose a communications protocol; each machine needs to be set to the same protocol so that each machine can understand the other's instructions. EZPATH supports three protocols: EZ-LINK, Ymodem, and ASCII.

If you wish to use Ymodem or ASCII data transfer, select 2 or 3, and skip to section 9.5.2.

EZ-LINK is a special communications protocol used by Bridgeport Machines equipment and software. Select 1 to use this protocol. The EZPATH system diskette contains a program called EZUTILS.EXE; it can be run on any remote MS-DOS computer (using COM1 only), and supports all three protocols.

9.5.1.1. EZ-LINK protocol

If you select EZ-LINK, you will need to specify some additional information. First, you must tell the communications software what kind of device you will be communicating with. When you select the EZ-LINK protocol, you will see screen shown in Figure 9-7.

```
EZ-Link CONNECTED To:

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

<ESC> QUIT.

>>> Select:
```

Figure 9-7 Using the EZ-LINK protocol

The options are 1, another EZPATH machine; 2, a computer running EZ-CAM software; or 3, a computer running EZ-FILE software.

EZ-LINK thinks of one device as the "master" device, and the other as the "slave." All of the commands to transfer files are issued at the master computer; the slave computer can only receive commands. EZ-LINK assumes that the EZPATH is the master and the other device is the slave—unless the other device *is* another EZPATH machine. Therefore, if you select 1 from the above menu, you will see the following menu (Figure 9-8), where you tell EZ-LINK whether your machine is the master or the slave:

```
EZ-Link MASTER \ SLAVE:

<1> MASTER.
<2> SLAVE.

<ESC> QUIT.

>>> Select:
```

Figure 9-8 If you are using EZ-LINK to transfer files, *and* the other machine is another EZPATH, you will need to specify whether your machine is the *master* device or the *slave* device.

Be sure you coordinate your settings with the other EZPATH machine.

9.5.2. Sending and receiving files

When all of your communication protocols have been set, you are ready to begin sending or receiving data, and you will see the menu shown in Figure 9-9. Note that if you are using EZ-LINK protocol, the slave device must be set on-line first, and the master device second. If the slave device is not yet on-line, do so now, before selecting one of the following commands.

Use option 1 to send a EZPATH (i.e., PGM) file to another device, and 2 to send a TXT file. Use 3 to receive a PGM file, and 4 to receive a TXT file. Use 5 and 6 to view directories of files of each type on your local EZPATH hard disk.

After selecting any of 1–4, your EZPATH machine will ask you for the name of the file to send or receive (as appropriate); if you are receiving the file, this is the name that it will be stored under on your hard disk. At this point, if you are receiving a file, the operator of the remote computer should select the file to be transmitted.

```
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.

>>> Select:
```

Figure 9-9 After setting your communications protocols, use these commands to send or receive a file.

Chapter 10
Math Help

10.1. Using the Calculator

Many EZPATH II operations require you to enter the coordinates of a point or location. To help you figure out the proper coordinates, EZPATH includes a Calculator. Whenever you are prompted to enter a set of coordinates, you can summon the Calculator by pressing the [* CALC] key. The Calculator screen is shown in Figure 10-1.

You can use the Calculator to do simple arithmetic (addition, subtraction, multiplication, division); exponents; trigonometry; or almost any algebraic expression combining these elements.

As you enter data in the Calculator window, it is displayed next to the **CALC:** indicator. Enter numbers and the arithmetic symbols +, –, /, and * with the numeric keypad on the right; you can group expressions by using the [← (] and [→)] keys to insert parentheses. The function keys F1–F6, and [↑ EXP] and [↓ SQRT], contain functions that require additional data from you (called *operands*). You will be prompted for this data by the **[ENTER] FUNC:** message. These functions are explained in detail below.

10.1.1. Sine

[F1 SIN]

This command computes the sine of an angle; at the **[ENTER] FUNC:** prompt, enter an angle (in degrees).

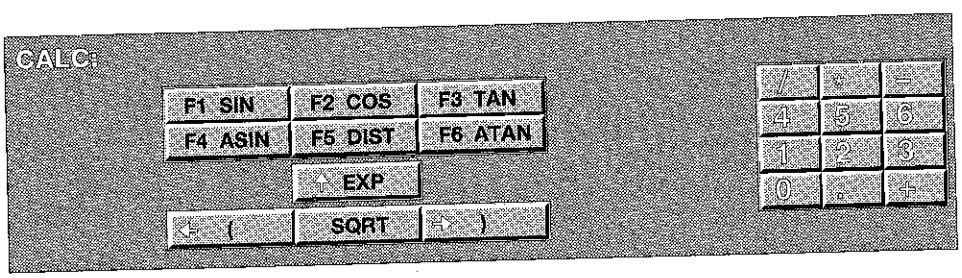


Figure 10-1 The EZPATH Calculator. Use the keypad on the right to enter numbers and basic arithmetic operators just like any calculator; hit [ENTER] when you are finished.

10.1.2. Cosine

F2 COS

This command enters the cosine function in the equation field. Again, it will prompt you for an angle (in degrees).

10.1.3. Tangent

F3 TAN

This command enters the tangent function in the equation field. At the prompt, enter an angle in degrees. If you enter 90 or -90 degrees, you will get unpredictable data, since the tangents for those values are undefined.

10.1.4. Arcsine

F4 ASIN

This command enters the arcsine function in the equation field. The number that you enter at the prompt must be between -1 and 1; otherwise, an error is returned by the system, and zero is entered in the coordinate field. The arcsine function lets you compute an angle in a right triangle if you know the value of one of the sides and the *hypotenuse* (the side opposite the right angle). In the example shown in Figure 10-2, divide the side opposite angle A by the hypotenuse, and enter this value at the [ENTER] FUNC: prompt.

10.1.5. Distance

F5 DIST

This function will prompt you for two values; hit ENTER after typing the first value, and it will prompt you for the second. If you visualize the two numbers as representing the lengths of the sides of a right triangle, then the value this function returns is the length of the third side (the *hypotenuse*). For example, if you enter 3 and 4, the Distance function returns a value of 5.

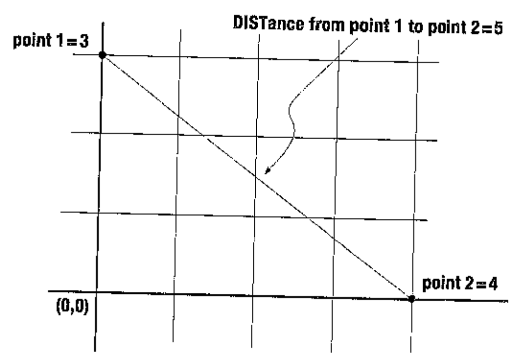


Figure 10-2 Using the Distance function.

10.1.6. Arctangent

F6 ATAN

This command enters the arctangent function in the equation field. The arctangent function calculates the measure of an angle in a right triangle by using the lengths of the 2 non-hypotenuse sides. The side *opposite* the angle you wish to compute should be entered first. For example, if the entered equation is ATAN(3,4), then the calculated value of 36.8966 is the measure of the angle *opposite* the side of length 3; see Figure 10-3.

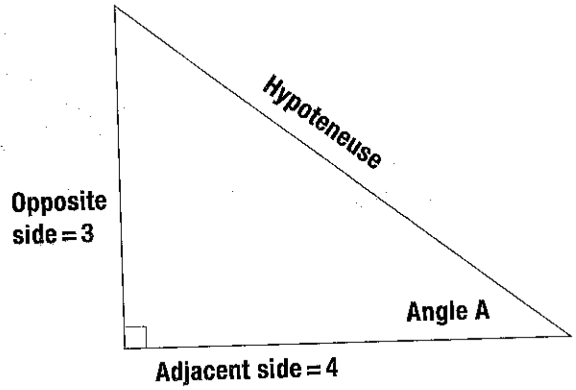


Figure 10-3 When you know the lengths of two (non-hypotenuse) sides of a right triangle, you can use the Arctangent function to solve for the angle; if you know the hypotenuse and one other side, use Arcsine.

10.1.7. Square Root

↓ SQRT

This command enters the Square Root function in the equation field. If a negative number is entered, the system returns an error and the number entered in the coordinate field is zero.

10.1.8. Exponent

↑ EXP

This function is used to enter an exponent. The exponent must be a positive whole number; you will get wrong data if you try to enter a negative exponent, or an exponent with a decimal point or fraction.

10.2. Geometry Help

In the last chapter you saw how the EZPATH Calculator can help you with arithmetic and trigonometric calculations. EZPATH also has a Geometry Help module, which will provide solutions to 16 common geometric situations. Like the Calculator, it is available whenever you are prompted to enter coordinate data, and will automatically enter the calculated value in the appropriate data entry field. You can reach the Geometry Help by pressing the /GEOM key whenever you see it on the screen.

Figure 10-4 shows the Geometry Help screen. After each key name on the left is an abbreviated description of what the function will solve for, based on the data you provide as described in the brackets. For example, the function for key 3 solves for the centerpoint of an arc if you provide the coordinates of two points on the arc and the radius. Key 4 solves for the centerpoint if you know two

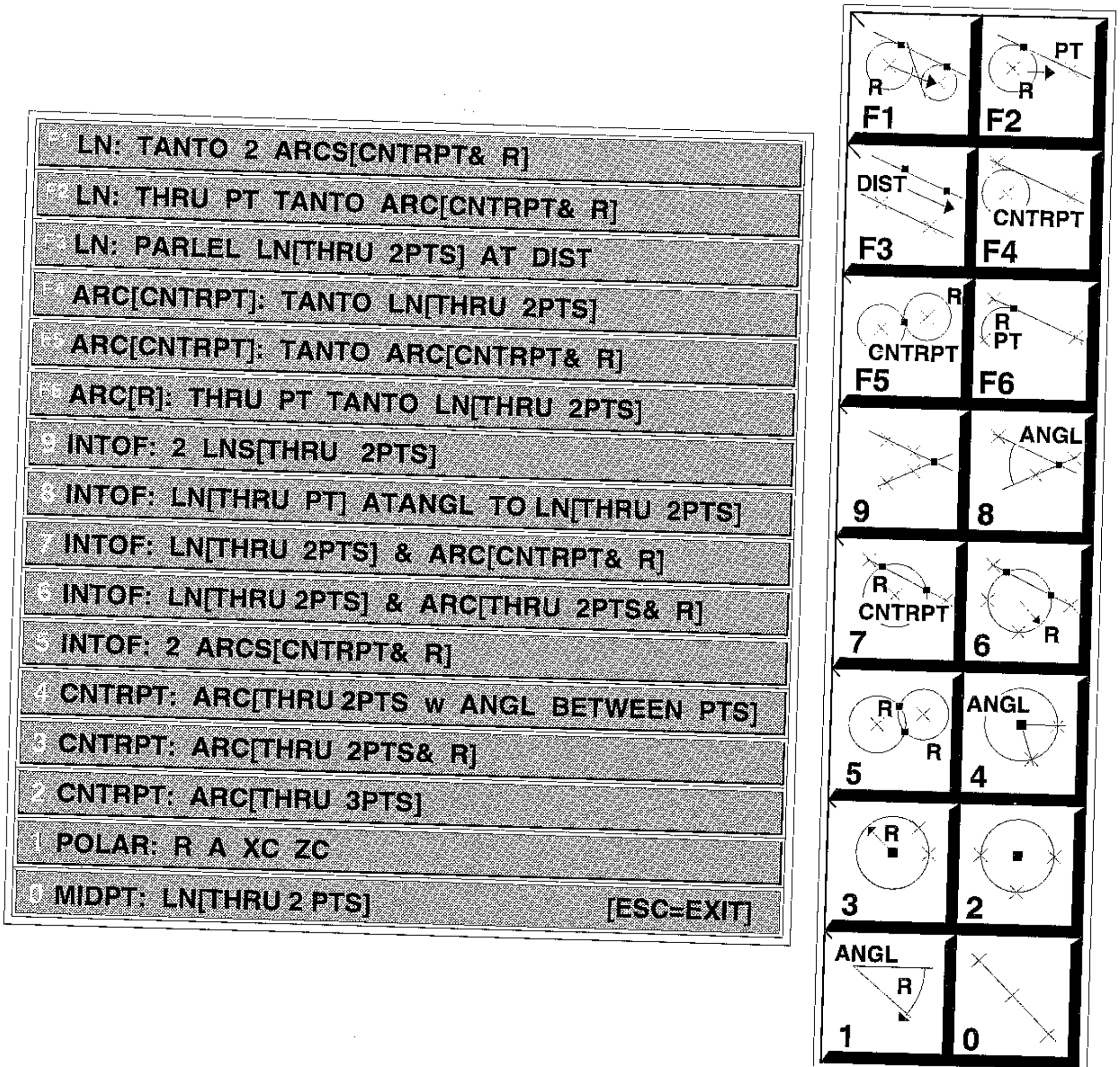


Figure 10-4 Geometry Help functions. On the left, next to the key names, is an abbreviation of what each function will solve for and, in brackets, what information you need to provide; on the right is a graphic summary of each key's problem.

points and the angle between them; key 2 will find it if you know three points on the arc. On the right side of the screen is a series of graphic depictions of each of these problems.

If a function has only a single point as an answer, EZPATH will automatically enter it into whatever command you were executing when you entered Geometry Help. If the function has 2 points for an answer, when it has calculated these points you will have the opportunity to choose which one you want to use.

The diagrams that accompany the descriptions of each function are drawn using a X-Z coordinate system, where X is the vertical axis and Z is horizontal. The coordinates of different points in the diagrams are listed as (x,z). For some of the functions, you will need to specify a direction, either left or right, so that EZPATH can calculate the correct set of data; in these cases, you will see an arrow labeled "LOOK," which you would use to select which direction is which.

For each function described below, there will be a screen illustration with sample data input, and next to it will be a diagram of that data. The solid dots with coordinate labels correspond to the input data; the point or points to be solved for are shown with a hollow dot. Each of these drawings is drawn to the same scale and includes the two coordinate axes.

10.2.1. Line tangent to 2 arcs

F1 LN: TANTO 2 ARCS[CNTRPT& R]

Solves for: Two points on a line which is tangent to two arcs

Input data: The center points and radius of each arc

There are four different lines that can be described as tangent to any two arcs, as illustrated in Figure 10-5. You will specify which of these lines you wish to solve for; EZPATH will return the coordinates of the two points where the line is tangent to each arc.

For each arc, you will need to specify the X- and Z-coordinates of its centerpoint, and its radius, as shown in Figure 10-6. Then, in two steps, you will specify which of the four lines you are solving for: first, you will specify either the inner pair (which intersect each other), or the outer pair (which do not); then, you will specify either the right or left line in the pair. Select left or right based on looking in the direction indicated by the arrow. Enter 1 to select the inner pair, and 0 to select the outer; then 1 to select the left line, or 2 to select the right one.

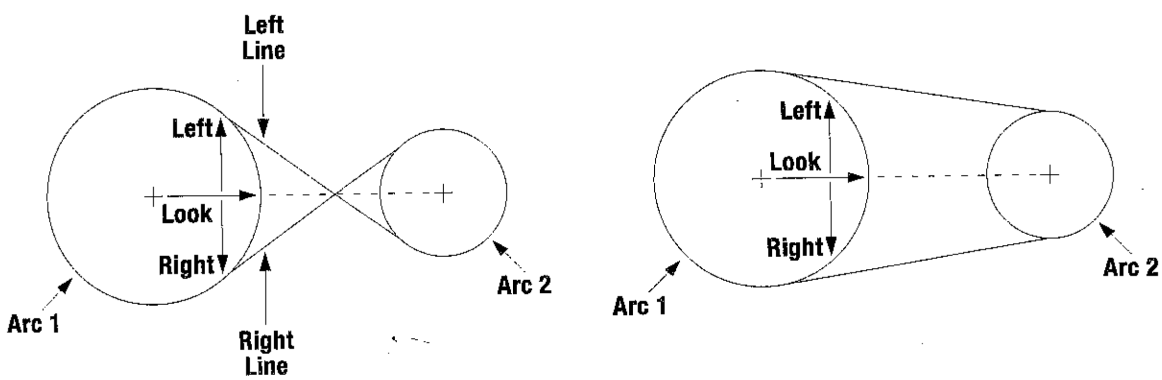
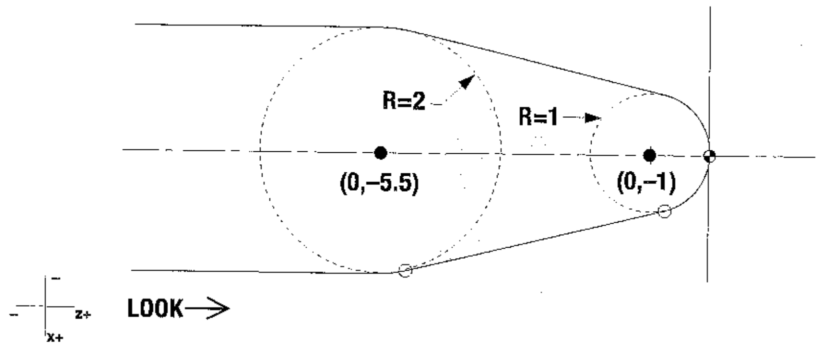


Figure 10-5 There are four lines tangent to any two (non-intersecting) arcs. The inner pair of lines cross each other *(right)*, while the outer pair do not *(left)*. To specify which line you want to solve for, after entering the coordinates of the two arcs, you will select one pair of lines as shown above; then, using the LOOK arrow, you will select the left or right line.



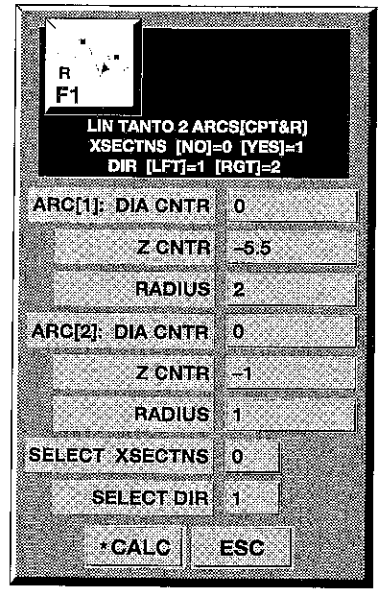


Figure 10-6 The data shown on the screen specifies that we are looking for the left line, in the non-intersecting pair.

EZPATH will calculate two points as shown in Figure 10-7; type 1 or 2 to select the one you want EZPATH to use. When you hit the ENTER key, EZPATH will automatically enter these coordinates into whatever command you were using when you called Geometry Help.

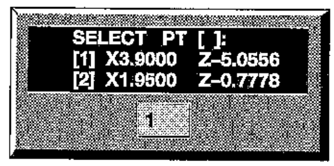


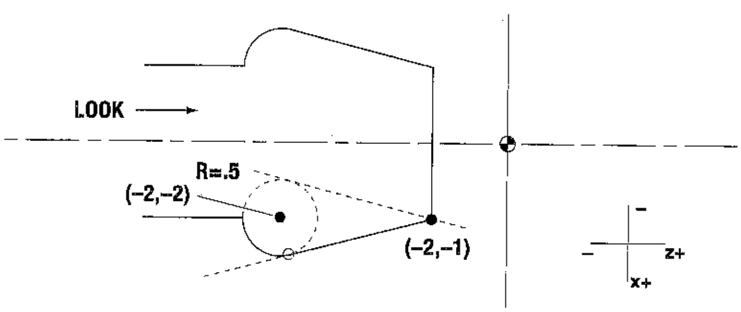
Figure 10-7 Select the point you are interested in.

10.2.2. Line tangent to 1 arc

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

Solves for: The tangent point of a line and an arc
Input data: One point on the line; the centerpoint and radius of the arc

This command calculates the point at which a line is tangent to an arc. As shown in Figure 10-8, for any point you might specify, there will be two lines through it which are tangent to the arc; you will be asked to select one. Enter 1 to select the left line, and 2 to select the right. In the example shown, the left line is selected.



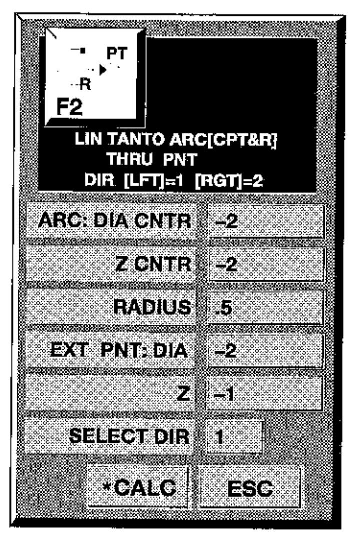
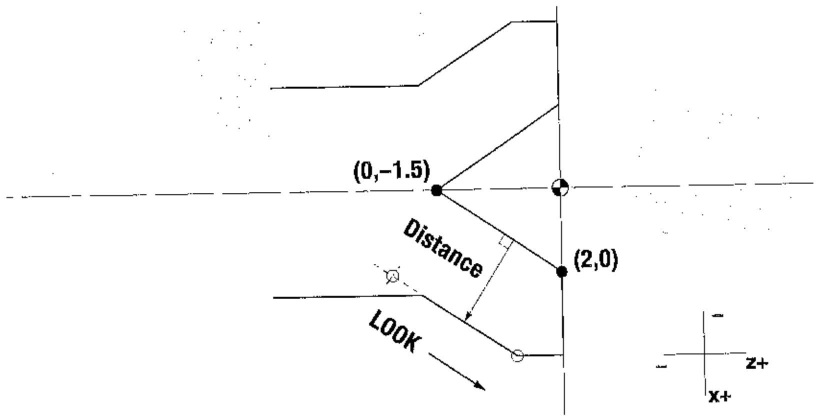


Figure 10-8 The F2 function finds where a line through a given point point is tangent to a particular arc. There are two possible tangent lines; this example specifies the left line.



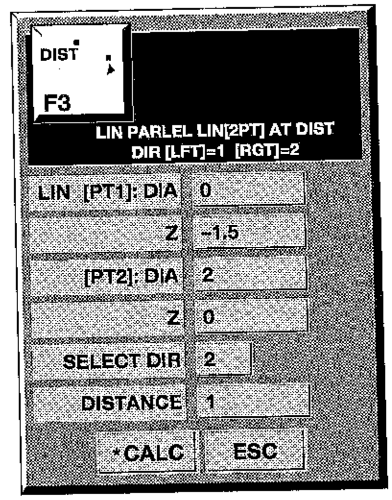


Figure 10-9 The F3 function gives you the coordinates of two points on a line either to the left or to the right of an existing line, and at a specified distance from it.

10.2.3. Parallel line

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

Solves for: Two points on a line which is a specified distance from an existing line

Input data: Two points on the current line; the distance from this line to the new one

For each of the two points you specify, EZPATH will calculate the point which lies at the specified distance perpendicular to the original line (see Figure 10-9) These two new points form a line parallel to the original one. You can use this function to calculate either of two parallel lines; in SELECT DIR, enter 1 to select the left line, and 2 to select the right. Afterwards, it will prompt you for which of the two points you want to use (Figure 10-10).

Note that the LOOK direction arrow is defined as the direction from the first point you enter to the second one.

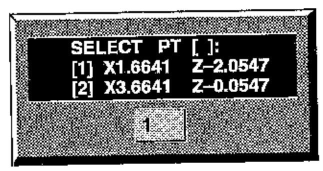
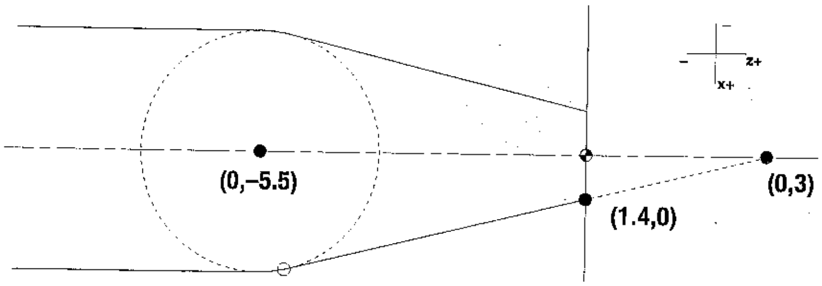


Figure 10-10 Select the point you wish to use by typing 1 or 2.



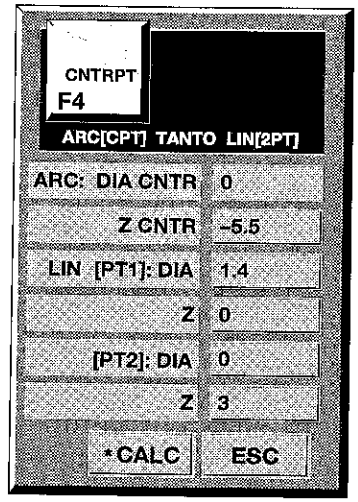


Figure 10-11 The F4 function solves for the tangent point of a line and an arc, based on two points on the line and the centerpoint of the arc.

10.2.4. Arc tangent to line

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

Solves for: The tangent point of a line and an arc
Input data: Two points on the line; the centerpoint of the arc

This function finds the same point as F2: the tangent point of a line and an arc. However, here you can find it based on different information. With this function, you do not need to know the radius of the arc; but, you need two points on the line, instead of just one. With this information, there is only one possible tangent line, so you do not need to choose a direction; see Figure 10-11.

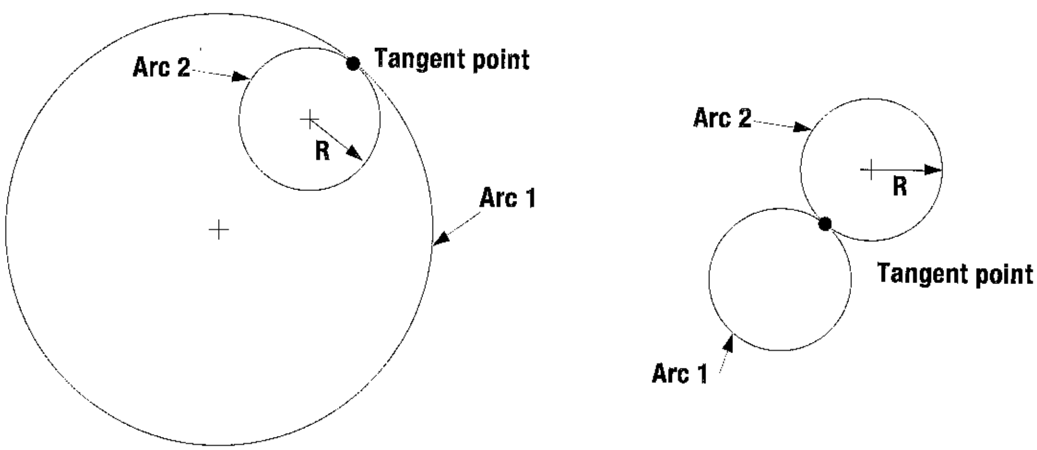


Figure 10-12 The tangent point between two arcs can be either *outside* the two center points (left drawing), or *inside* the two center points (right drawing).

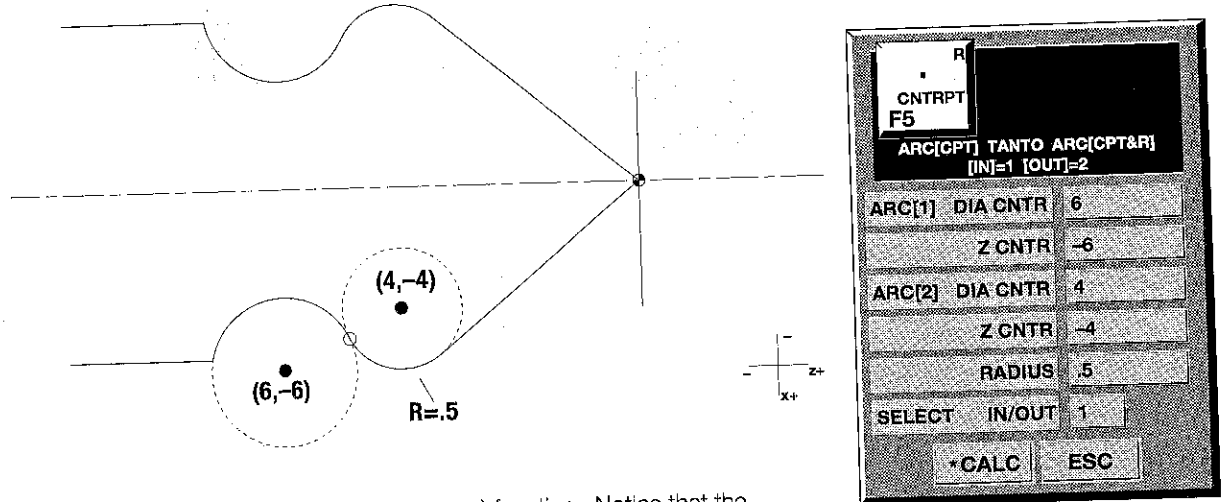


Figure 10-13 The F5 (tangent point of two arcs) function. Notice that the tangent point will always be on a straight line between the two centerpoints; after finding the tangent point with the F5 function, you could easily find the radius of arc 1 as well.

10.2.5. Arc tangent to arc

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

Solves for: The tangent point of two arcs

Input data: The centerpoints of both arcs; the radius of the second arc.

This function finds the tangent point between two arcs. With the type of data that you provide, there are two possible tangent points: a tangent point between the two centerpoints, or a tangent point outside the two centerpoints. Figure 10-12 illustrates the difference. The example shown in Figure 10-13 solves for an inside tangent point.

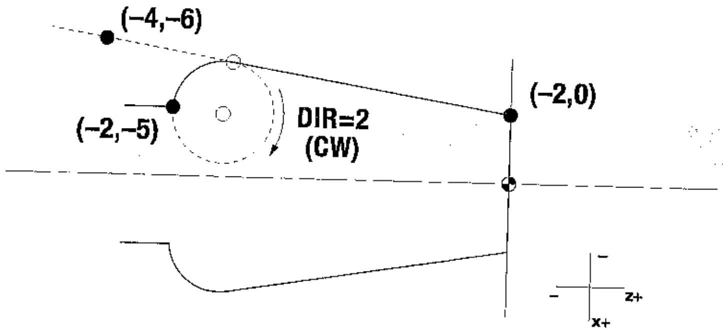
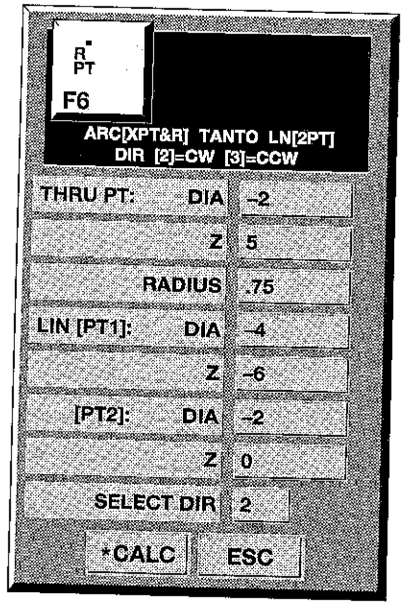


Figure 10-14 The solid-line arc, used in the sample data at the right, illustrates the clockwise direction; the dotted-line arc is counterclockwise.



10.2.6. Arc tangent to line

F6 ARC[R]: THRU PT TANTO LN[THRU 2PTS]

Solves for: The tangent point of an arc and a line; *and*, the center point of the arc
Input data: A point on the arc; its radius; two points on the line

Like F2 and F4, you can use this function to find the tangent point of an arc and a line; in this case, besides needing two points on the line, you need one point on the arc and its radius. In addition to the tangent point, this function will also tell you the centerpoint of the arc.

Based on the data that you input, there can be two possible arcs: you can move from the arc-point towards the line in either the clockwise (CW) or counterclockwise (CCW) direction. It's easier to understand if you look at Figure 10-14: the solid line is a clockwise arc, as used in the example; the counterclockwise arc is shown by the dotted line. In the DIR field, enter 2 for clockwise, or 3 for counterclockwise.

After you enter your data, the function will return the coordinates of the tangent point and the centerpoint; select the one you wish to use as shown in Figure 10-15.

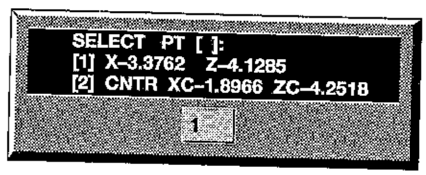
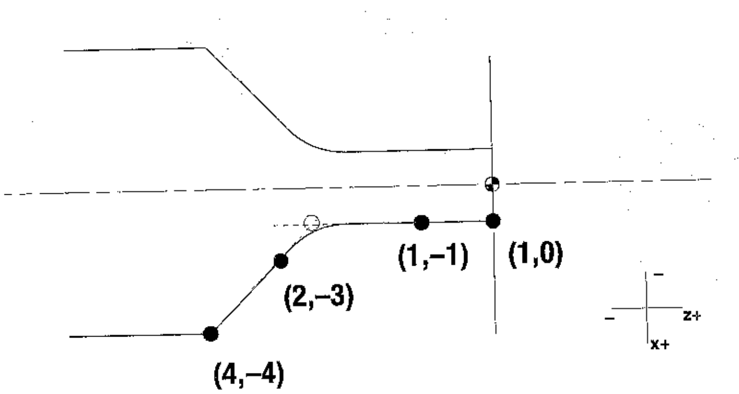


Figure 10-15 Enter 1 to select the tangent point; enter 2 to select the centerpoint of the arc.



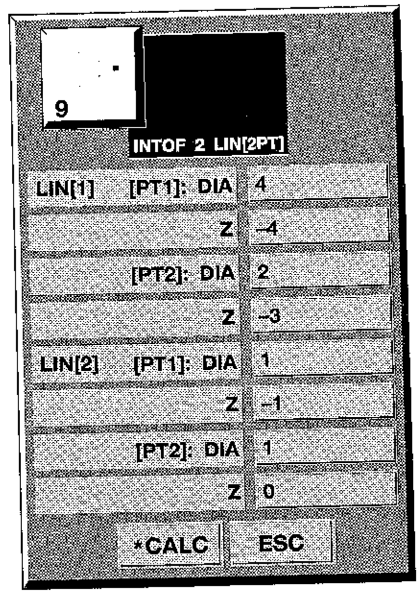


Figure 10-16 Use function 9 to find the intersection of two lines when you already know two points on each line.

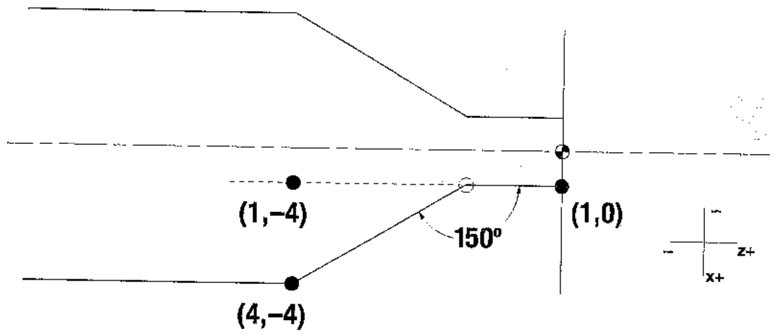
10.2.7. Intersection of 2 lines

9 INTOF: 2 LNS[THRU 2 PTS]

Solves for: The intersection point of two lines

Input data: Two points on each line

EZPATH gives you two functions to calculate the intersection point of two lines. For this one, you need to specify two points on each line, as shown in Figure 10-16.



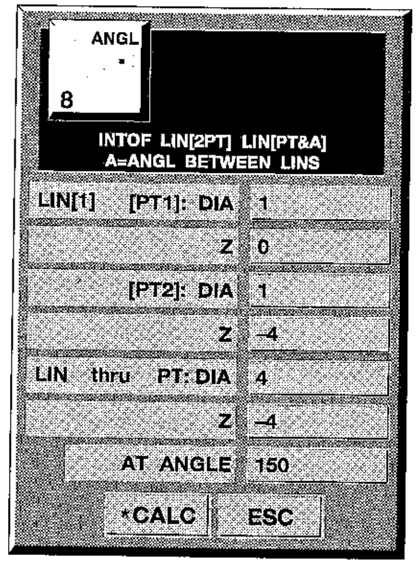


Figure 10-17 You can also use this function to find the intersection point of two lines; in this case, for the second line, you would need the coordinates of 1 point and the angle between the two lines. Use degrees to specify the angle.

10.2.8. Intersection of line at angle to line

8 INTOF: LN[THRU PT] ATANGL TO LN[THRU 2PTS]

Solves for: The intersection point of two lines

Input data: Two points on the first line; one point on the second line; the angle between the two lines.

Like function 9, this calculates the intersection point of two lines, but instead of a second point on line 2, it uses the angle between the two lines, as shown in Figure 10-17. To properly specify the angle, measure the angle clockwise from line 2 to line 1.

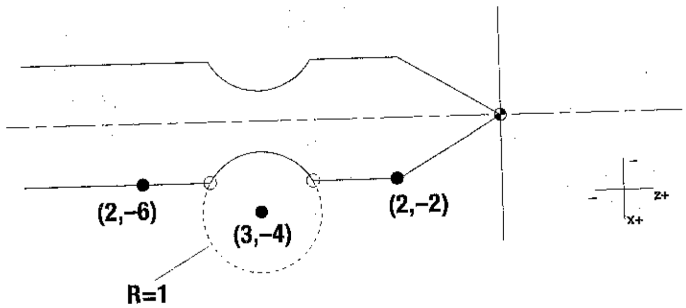
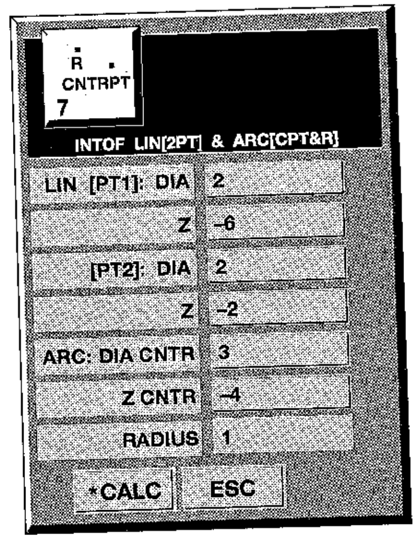


Figure 10-18 Function 7 calculates the two points where a line running through an arc intersects it.



10.2.9. Intersection of line and arc

7 INTOF: LN[THRU 2PTS] & ARC[CNTRPT& R]

Solves for: The intersection points of a line through an arc

Input data: Two points on the line; the centerpoint and radius of the arc

This function calculates the two points where a line intersects an arc. Notice the difference between this function and the functions of F2, F4, or F6: those functions calculate the tangent point of a line and an arc, where the line runs outside the arc and just touches it in one point. Use this function where the line runs through the arc and intersects it in two points. See Figure 10-18. After it calculates the two points, it will give you the opportunity to select the one you wish to use (Figure 10-19).

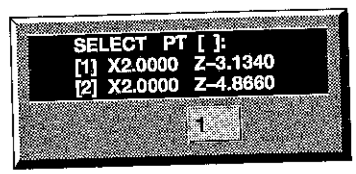


Figure 10-19 Select the point you wish to use

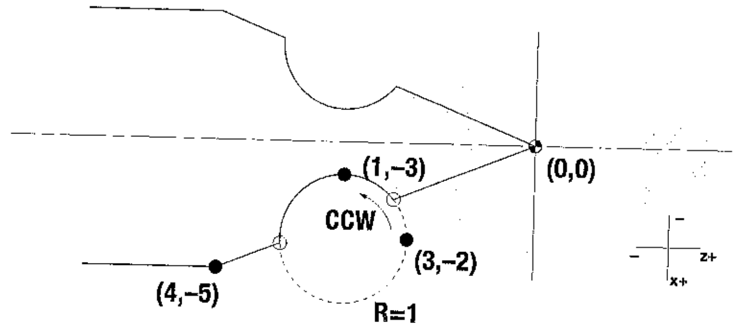
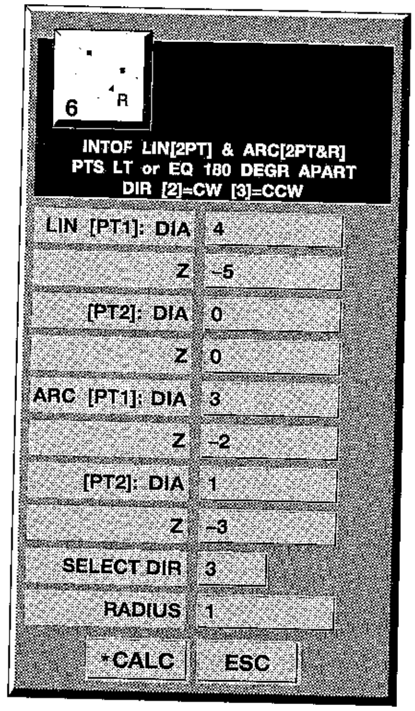


Figure 10-20 Like function 7, function 6 calculates the intersection points of a line and an arc; however, it uses two points on the arc, instead of the centerpoint. Note that you also need to specify a direction for the arc: clockwise=2, counterclockwise=3.



10.2.10. Intersection of line and arc

6 INTOF: LN[THRU 2PTS] & ARC[THRU 2PTS& R]

Solves for: The intersection points of a line through an arc

Input data: Two points on the line; two points on the arc; the radius of the arc

This function calculates the same points as the previous function; use this one where you don't know the centerpoint of the arc, but do know the coordinates of two points on the arc.

Unlike the previous function, here you will need to specify a direction for the arc, either clockwise (2) or counterclockwise (3); see Figure 10-20. The proper direction is determined by moving from Point 1 to Point 2 along the arc, so you need to be careful that you enter the points in the proper order. Like the previous function, you will need to select the point you want from the two that it will calculate (Figure 10-21).

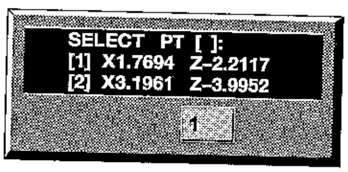
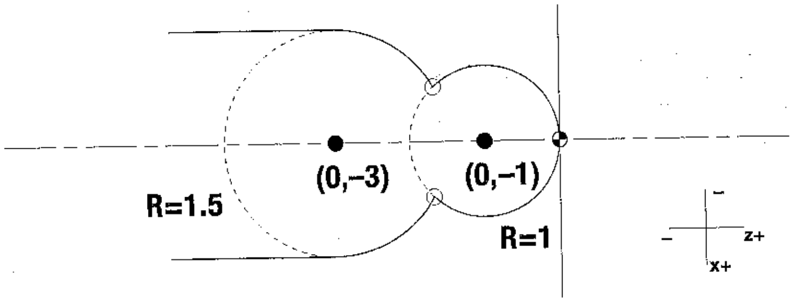


Figure 10-21 The intersection points calculated from Figure 10-20.



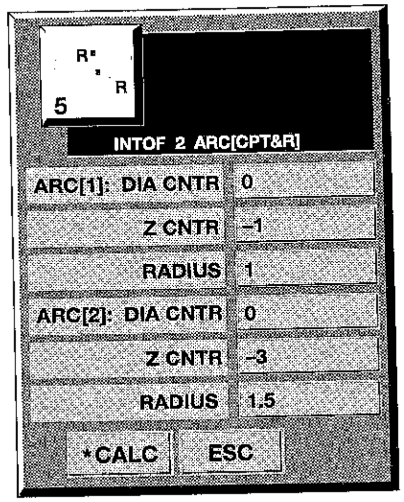


Figure 10-22 Use function 5 to solve for the two points where two overlapping arcs intersect.

10.2.11. Intersection of 2 arcs

5 INTOF: 2 ARCS[CNTRPT& R]

Solves for: The intersection points of two arcs

Input data: The centerpoint and radius of each arc

This function calculates the points where two overlapping arcs intersect (Figure 10-22); for each arc, you need to specify the radius and centerpoint. After it calculates the two points, it will give you the opportunity to select the one you wish to use (Figure 10-23).

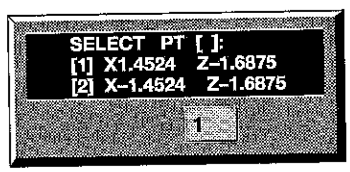
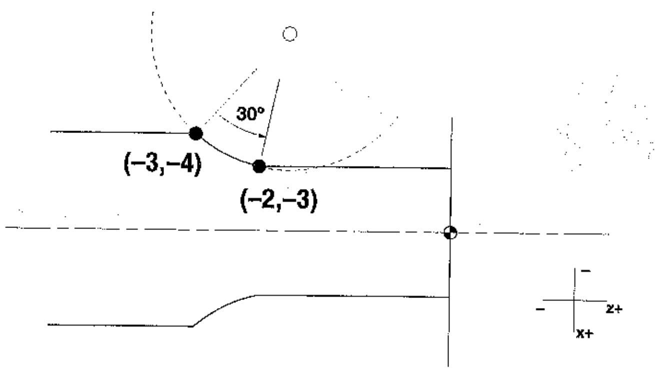


Figure 10-23 The two points calculated by function 5. Iif the two arcs meet at only one point (i.e. are tangent to one another) the coordinates for point 1 and point 2 will be the same.



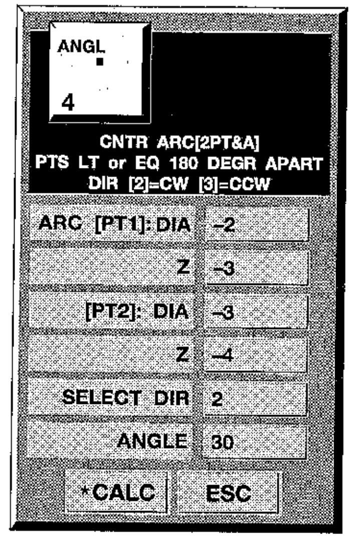


Figure 10-24 Functions 4, 3, and 2 all solve for the centerpoint of an arc. Use function 4 when you know two points and the angle between them.

10.2.12. Centerpoint of an arc

4 CNTRPT: ARC[THRU 2PTS W ANGL BETWEEN PTS]

Solves for: The centerpoint of an arc

Input data: Two points on the arc and the angle between them

This is the first of three functions that will solve for the centerpoint of an arc. For this function, you need to specify two points on the arc, and the angle between them (see Figure 10-24). Specify the angle in degrees. You will need to specify a direction for the arc as well: either clockwise (2) or counterclockwise (3), determined by the direction you are traveling if you move along the arc from point 1 to point 2.

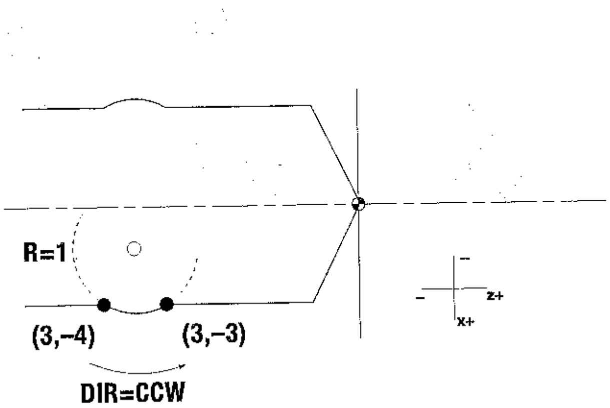
10.2.13. Centerpoint of an arc

3 CNTRPT: ARC[THRU 2PTS& R]

Solves for: The centerpoint of an arc

Input data: Two points on the arc and the radius

This function works exactly the same as the previous one, except that instead of specifying the angle between the points you need to know the radius of the arc. Again, you need to specify a direction as you move from point 1 to point 2 along the arc: either clockwise (2) or counterclockwise (3). See Figure 10-25.



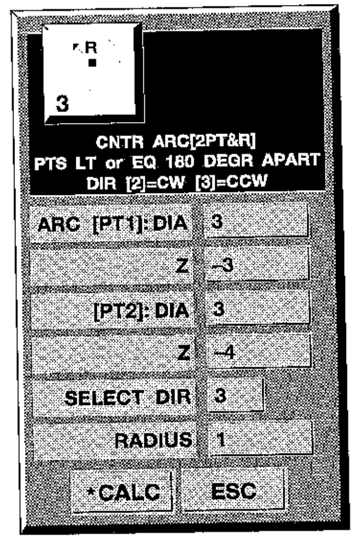


Figure 10-25 Use function 3 when you know two points and the radius. Functions 4 and 3 each require you to enter a direction for the arc, either clockwise or counterclockwise.

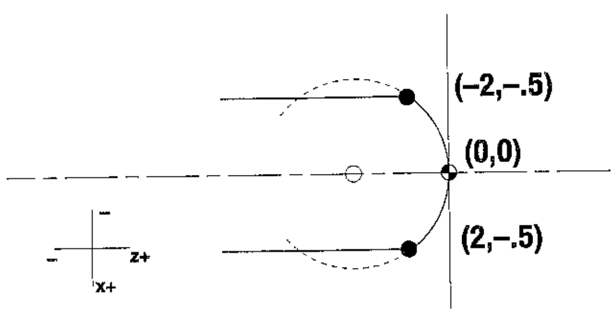
10.2.14. Centerpoint of an arc

2 CNTRPT: ARC[THRU 3PTS]

Solves for: The centerpoint of an arc

Input data: Three points on the arc

This function finds the centerpoint of an arc based on the coordinates of three points on the arc. There is no need to specify a direction for this function, since any three points can have only one arc in common. See Figure 10-26.



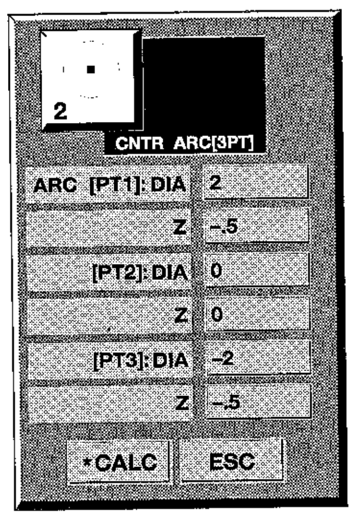
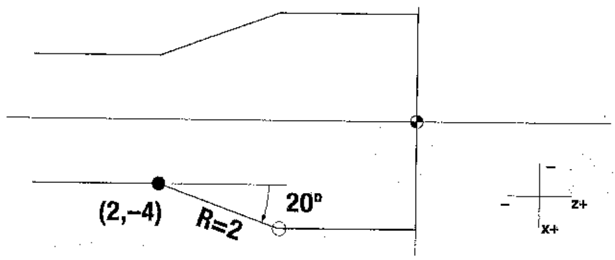


Figure 10-26 Use function 2 when you know three points on the arc.



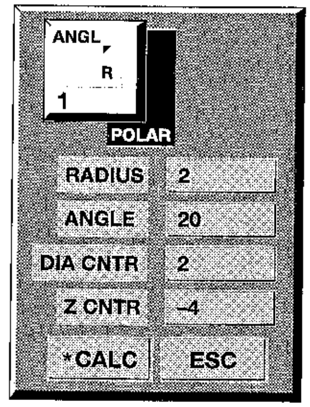


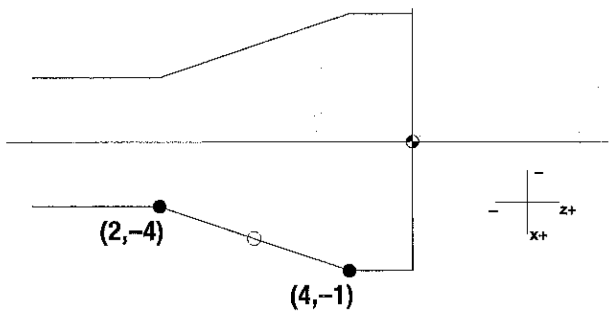
Figure 10-27 Function 1 converts from polar to rectangular coordinates. Notice that these polar coordinates are measured not from point (0,0) but from (1,2); you can specify this point (called a *pole*) in X CNTR, Z CNTR in the box at right.

10.2.15. Polar coordinates

1 POLAR: R A XC ZC

Solves for: X-Z coordinates of a point
Input data: Polar coordinates of a point

Use this function to convert polar coordinates to rectangular (X-Z) coordinates. Polar coordinates specify a point in terms of the distance from an origin (radius) and the angle from the horizontal; in the example shown in Figure 10-27, the hollow point is 2 units from the origin, at an angle of 30° from the horizontal. Use this function to convert these to X- and Z-axis values. If the origin for your polar coordinate system (the *pole* location) is not point (0,0), you need to enter its X-Z coordinates in the X CNTR and Z CNTR boxes; for example, in Figure 10-27, the polar coordinates are measured from point (2,–4).



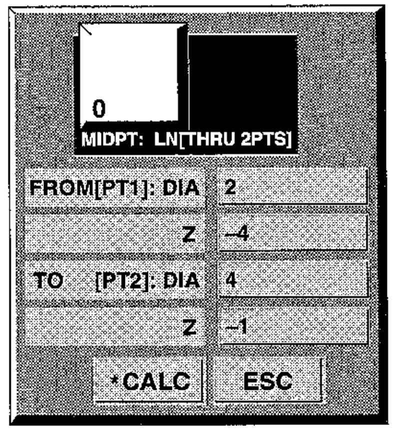


Figure 10-28 Function 0 finds the midpoint of a line segment if you specify the two endpoints.

10.2.16. Midpoint of a line

0 MIDPT: LN[THRU 2PTS]

Solves for: The midpoint of a line segment
Input data: Two endpoints

If you enter the coordinates of two points, this function will return the coordinates of the midpoint of a line segment between them, as shown in Figure 10-28.

Index

Note: References are to section numbers, not page numbers

D

E

F

Note: References are to section numbers, not page numbers

Note: References are to section numbers, not page numbers

Note: References are to section numbers, not page numbers

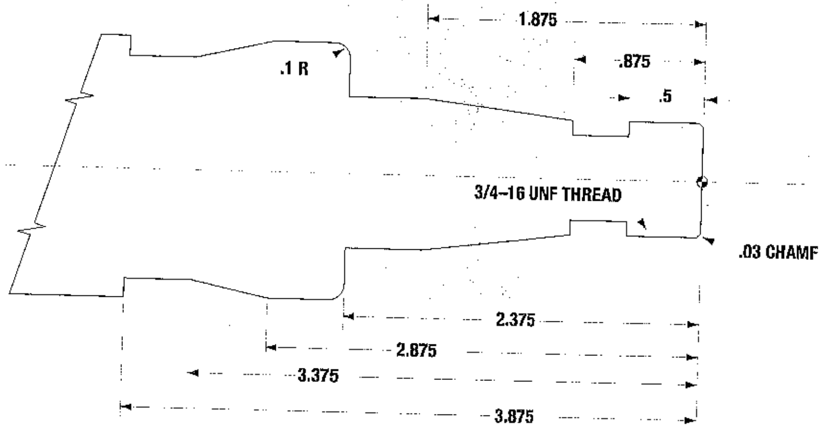
Note: References are to section numbers, not page numbers

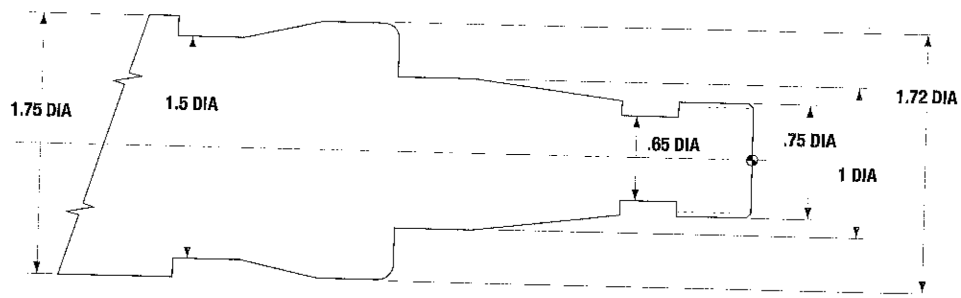
EZPATH II

Programming Workbook

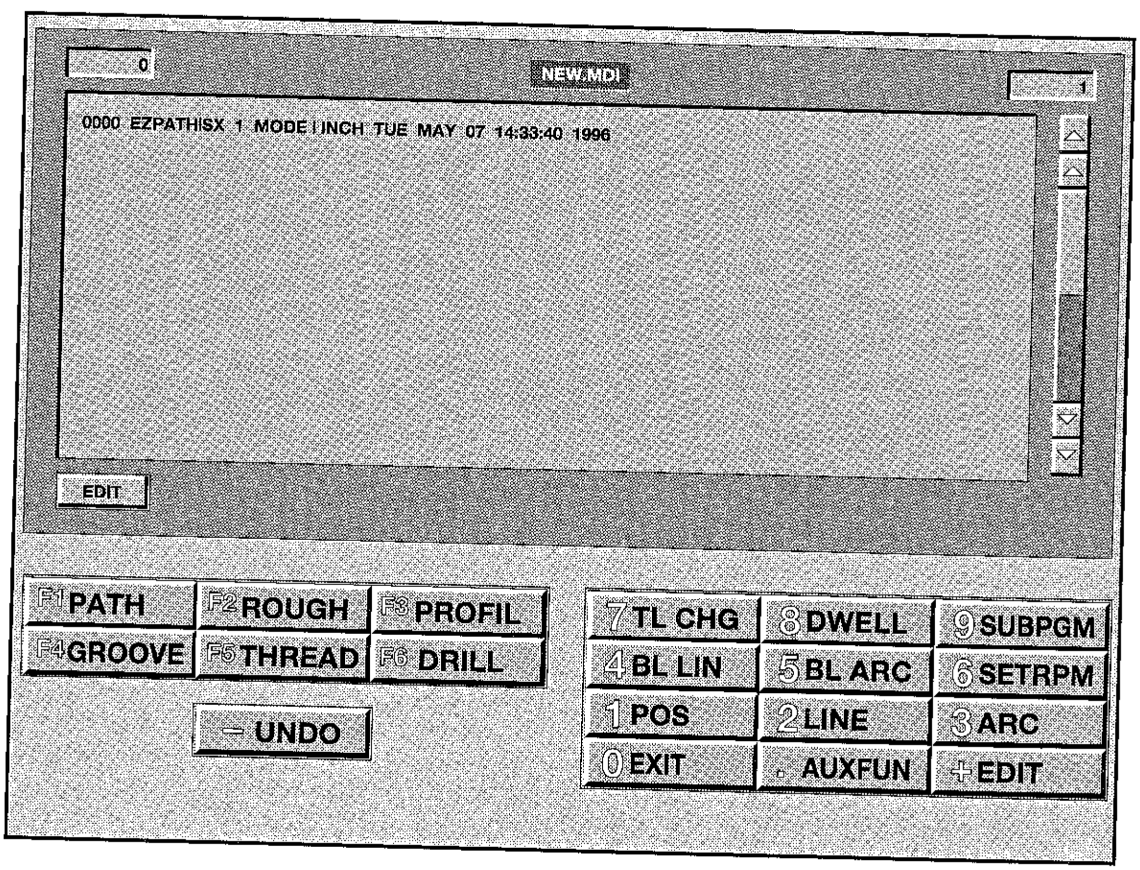
Bridgeport

Bridgeport Machines, Inc.
500 Lindley Street
Bridgeport, CT 06606





Above, part blueprint reference; ***below,*** MDI command screen



Programming in MDI

The following sections give you a line-by-line guide for creating an MDI program that will cut the part illustrated on the facing page. Each section includes a line from the program and the dialog box from the MDI command; use the drawings to fill in each box with the required information that will produce the program line. You can then recreate the program on your own machine by following the exact same steps. "Section" references are to the *EZPATH II Operations and Programming* manual; chapter 6 of that manual has a complete description of MDI programming, the various commands, and the canned cycles.

To actually run this program and cut the part, you would need three different tools: an OD turning tool; an OD grooving tool; and a 60° threading tool. *The tool IDs, numbers, and offsets used in this program are for illustration only: before trying to run this program on your machine, you must make the correct entries in your own Tool Library as described in section 3.1 of the EZPATH Programming manual, and set your offsets properly as described in section 3.2.* Use a 1.75" aluminum bar for your stock, secured in the chuck so that there is at least 4" hanging out.

WARNING!

To enter MDI mode, press the [8 MDI] key from the main EZPATH screen. Remember, if you make a mistake while entering the program data in MDI, you can use the [- UNDO] key to "back up" as many lines as you need to. If you do this and then re-enter the lines, the sequence numbers (i.e. line numbers) will be different than shown here, but that will not affect the program operation. Press the [0 EXIT] key after your program is finished to leave MDI and save your program on disk.

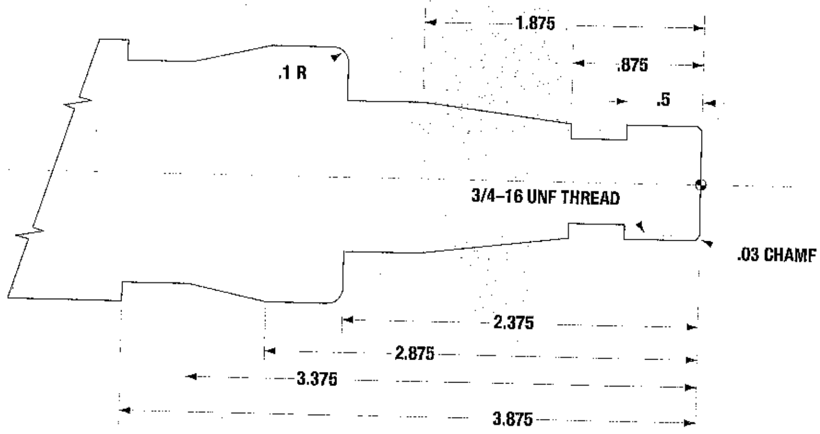
In following pages, lines written like this: **0010 STARTPATH 1** represent the actual program lines that EZPATH will create as you fill in the data for each command. For each such line, the first 4-digit number is called the *sequence number* (or line number); the rest of the words and numbers represent the numbers you type into the dialog boxes of each MDI command.

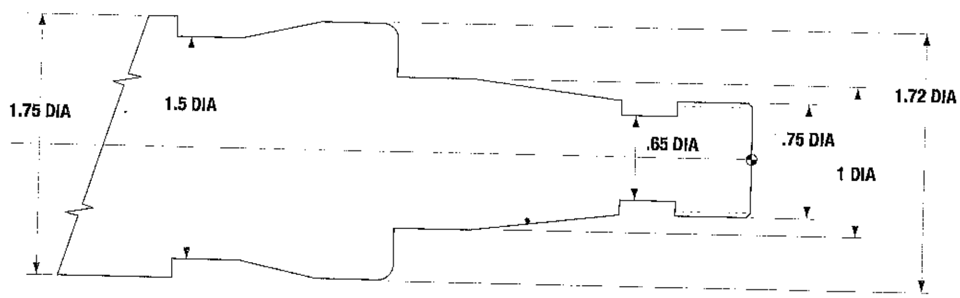
The entire program is listed separately after this tutorial.

[8 MDI]

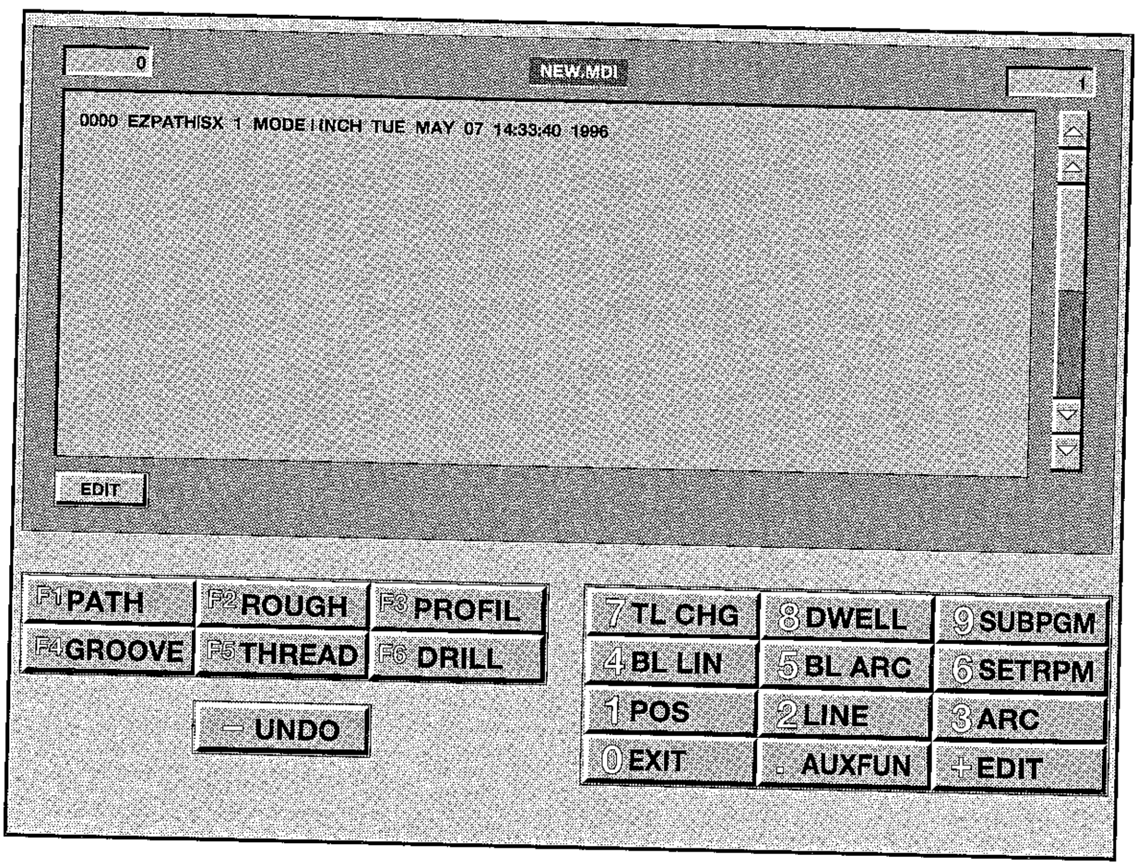
0000 EZPATH 1 MODE I INCH MON JUL 15 15:39:31 1996

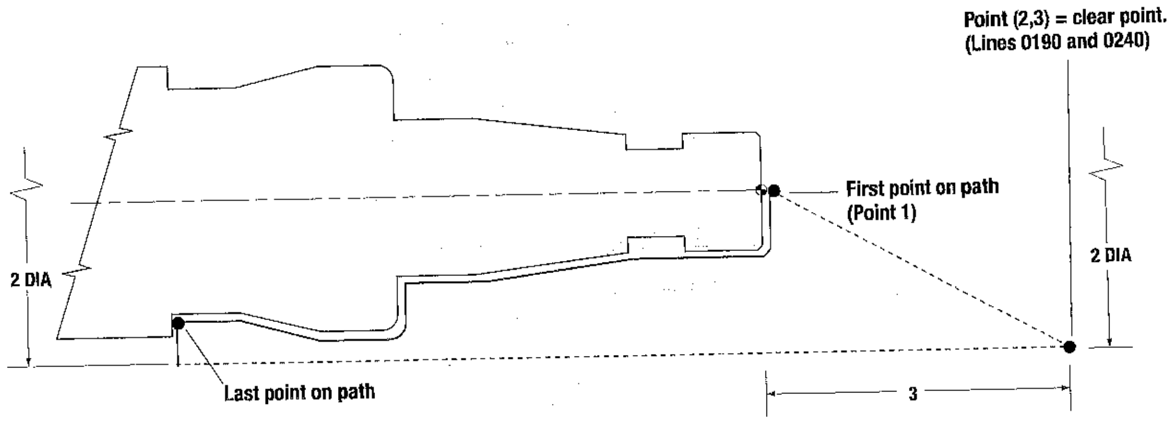
EZPATH will automatically create a line similar to this at the beginning of each program. You cannot edit or delete it. EZPATH automatically fills it in with the name of the machine; whether the program is in inches or millimeters; and the date and time it was created.





Above, part blueprint reference; ***below,*** MDI command screen

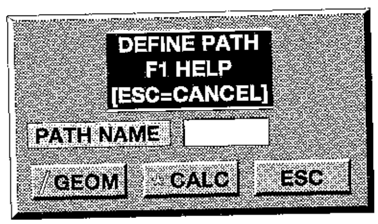




Creating the first path The heavy line shows the shape defined by the first path. The tool will make a rapid positioning move from the clear point on the far right; then, after the Roughing operation which uses the path, line 0230 moves the tool back to that same point. *(Note that the distances in this diagram are somewhat exaggerated.)*

F1 PATH

0010 STARTPATH 1



MDI command: Define Path

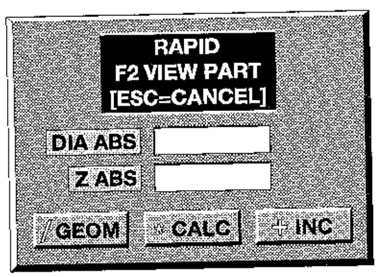
We'll begin the program with the 2 path definitions that together will describe the contour of the part for the cutting commands. The first path will cover all of the part except the inner-most groove.

To help you enter the proper data for each point on the path, at the top of each page will be a drawing of the path with the points numbered in order; on each facing page will be the properly dimensioned blueprint.

Fill in the dialog box with the path number.

1 POS

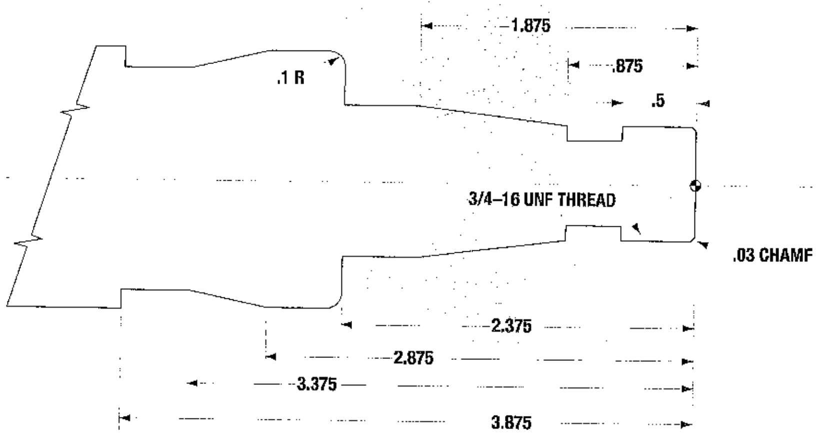
0020 RAPID ABS X0.0000 Z0.0000

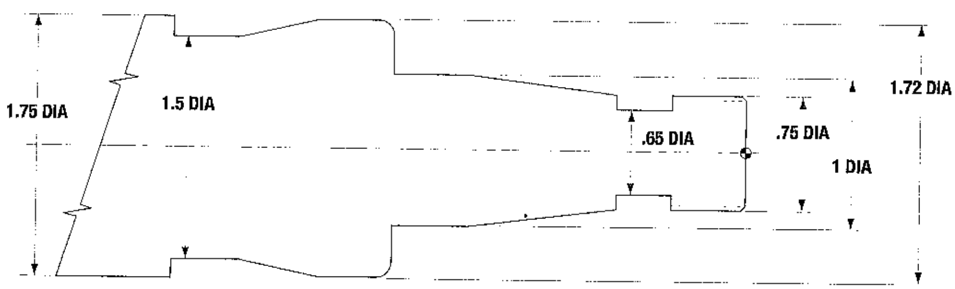


MDI command: Position

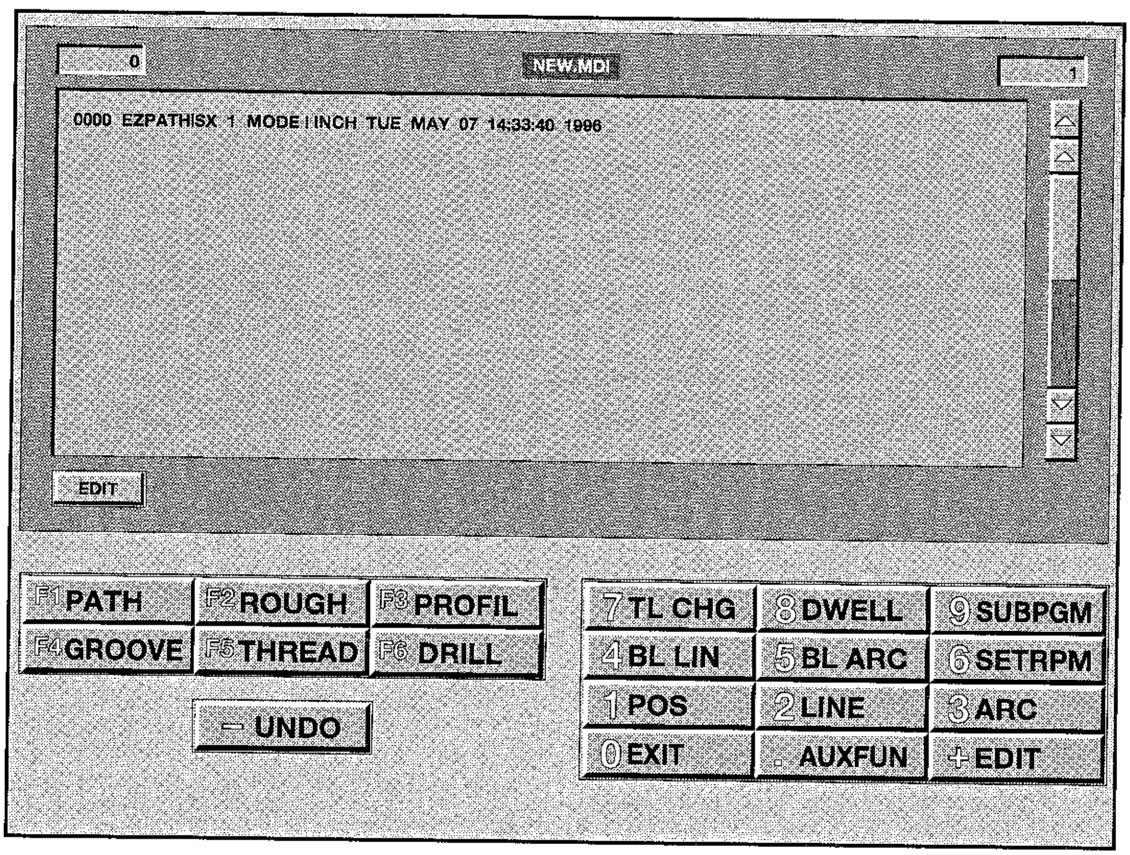
The first command in any Path definition must be a rapid positioning move to the first point of the path. Note that even though the tool will move to the beginning of the path as a rapid positioning move, you will specify a clearance distance when you select your canned cycle (in this program, line 0230), so it is OK to program the first point of the path definition right on the part.

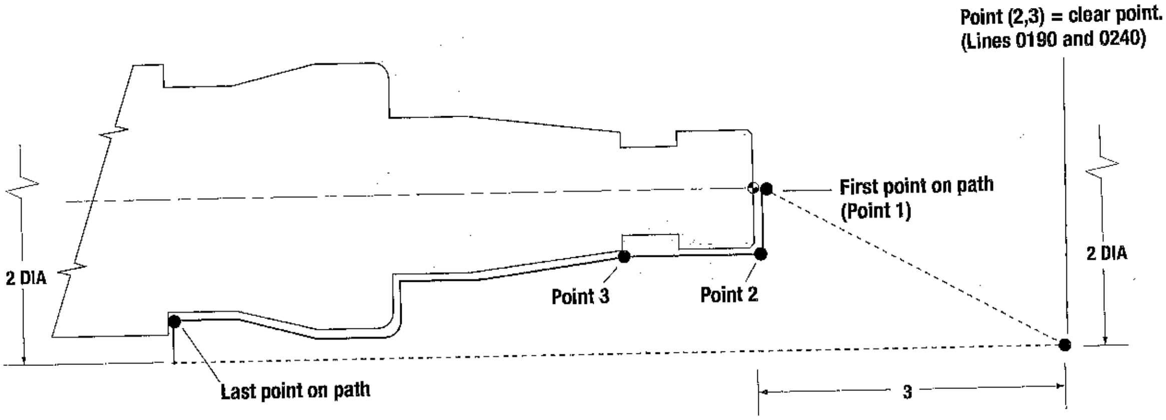
Fill in the dialog box with the coordinates of Point 1.





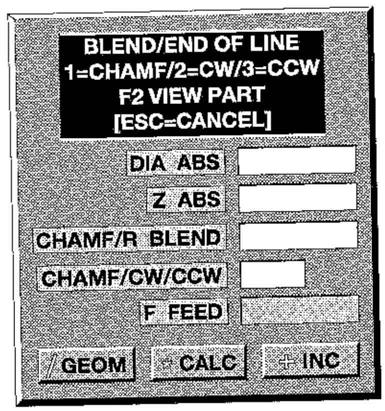
Above, part blueprint reference; ***below,*** MDI command screen





4 BL LIN

0030 CHAMFER ABS X0.7500 Z0.0000 P0.0300 P0.0300 F0.0100



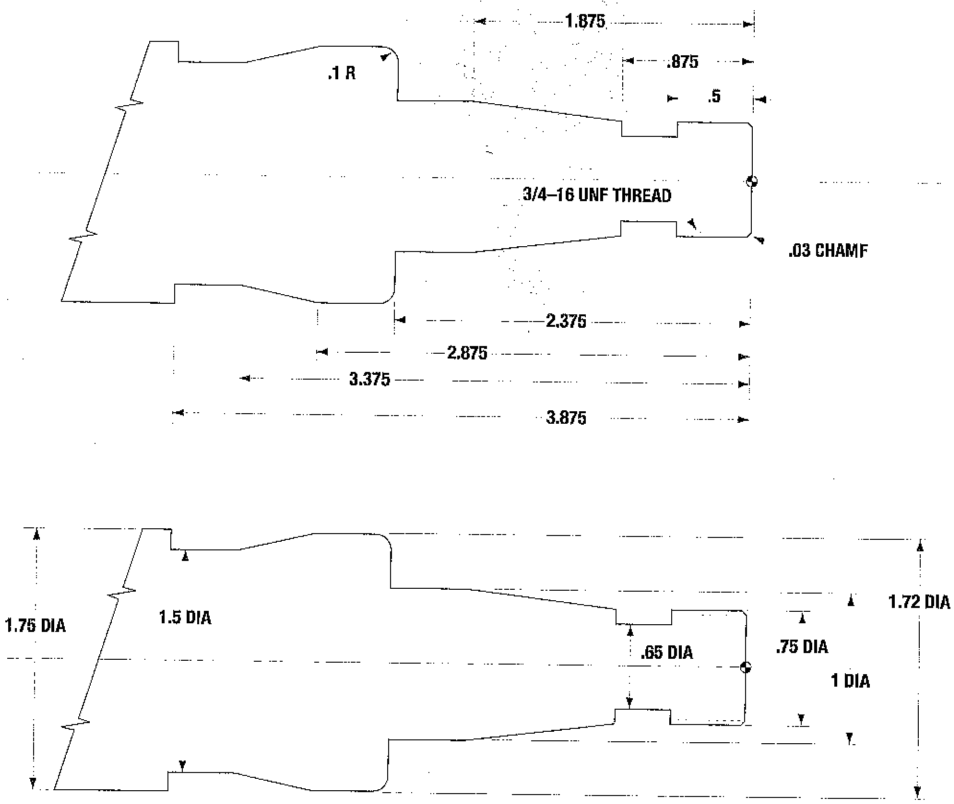
MDI command: Blend Line

Use the Blend Line command to cut along the front face of the part and then begin cutting the .03" chamfer. This command tells the EZPATH to begin cutting a straight line out to point (.75,0); however, it also calculates where it should cut a chamfer that will join this line with the line in the next statement, and before it gets to (.75,0), has already begun to cut that chamfer.

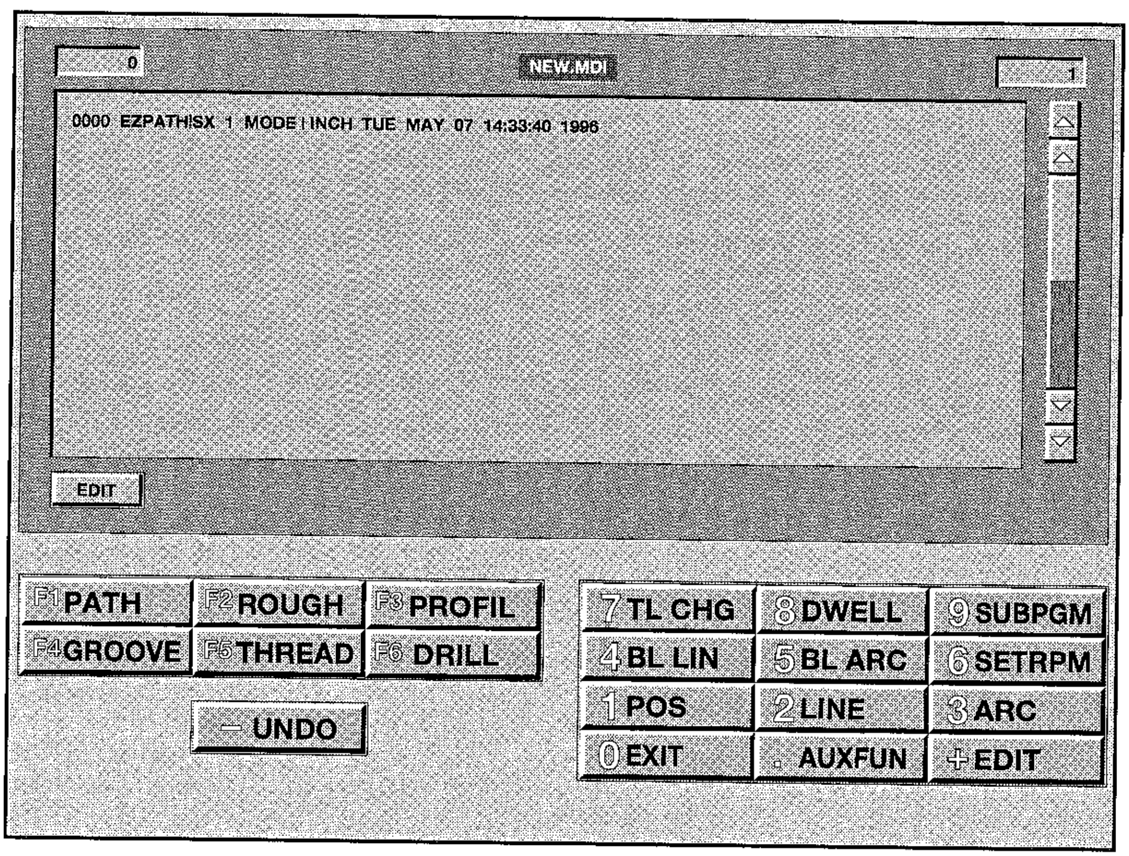
Notice that the F FEED box, where you would normally enter a feedrate, is inaccessible when you are defining a path; that is because the canned cycle which uses the path will include a feedrate. The feedrate listed in the program line is a "dummy" value.

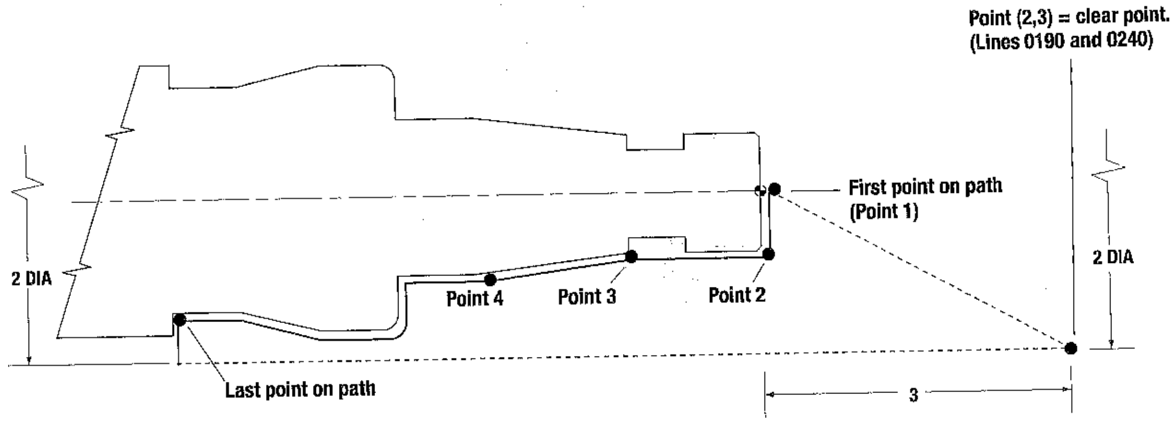
Keep in mind that this and the subsequent commands which comprise the path definitions are described as if the tool is making the cutting moves now, but this is of course not the case; while the EZPATH is reading these instructions, the tool is not doing anything at all. The control unit is simply building up in its memory the shape you're defining here. It will not begin cutting until a canned cycle calls the path; and even then, the tool motion will not necessarily be as described here, but will be based on the cutting motion particular to each cycle type.

Fill in the dialog box with the coordinates of Point 2 and the chamfer dimensions.



Above, part blueprint reference; ***below,*** MDI command screen



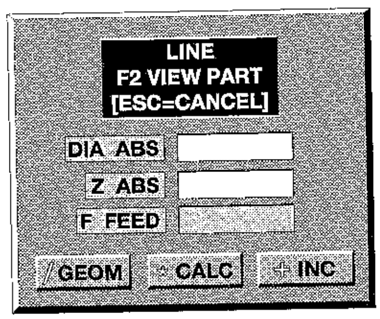


0040 LINE ABS X0.7500 Z-0.8750 F0.0100

MDI command: Line

After cutting the chamfer, it cuts on a straight line in the Z-axis back to point Z = –.875; the threads and groove will be cut later.

Fill in the dialog box with the coordinates of Point 3.

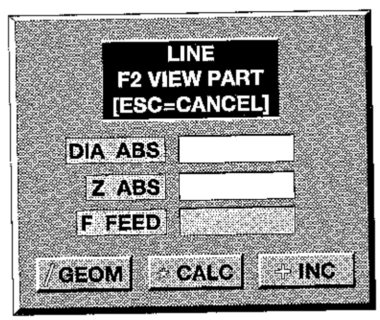


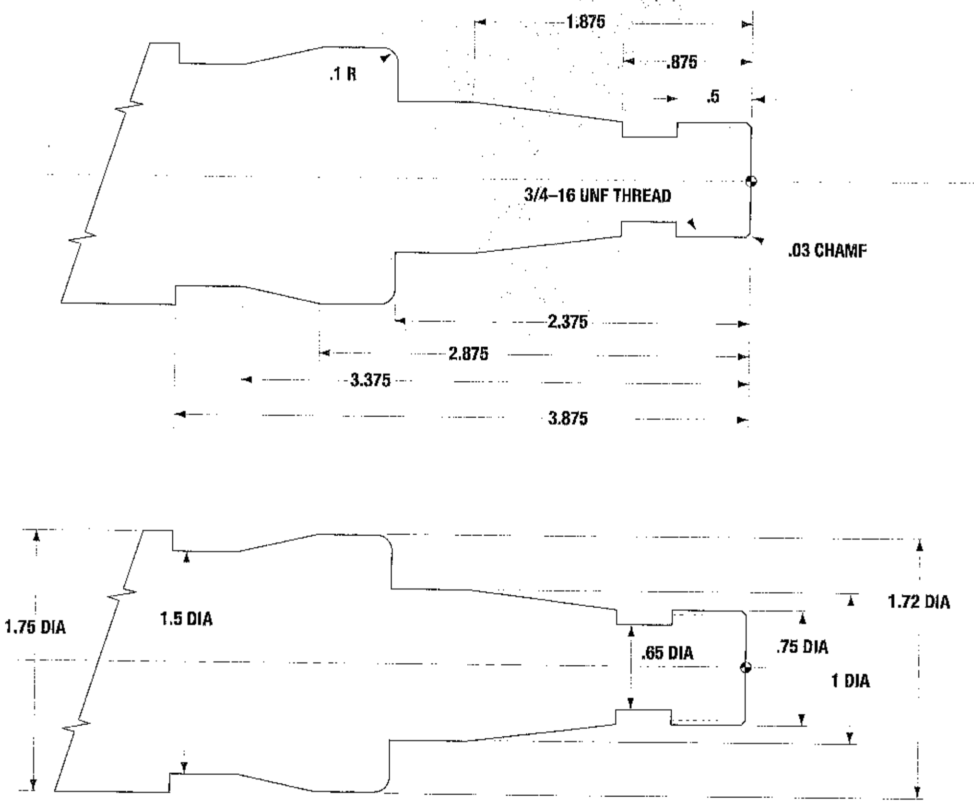
0050 LINE ABS X1.0000 Z-1.8750 F0.0100

MDI command: Line

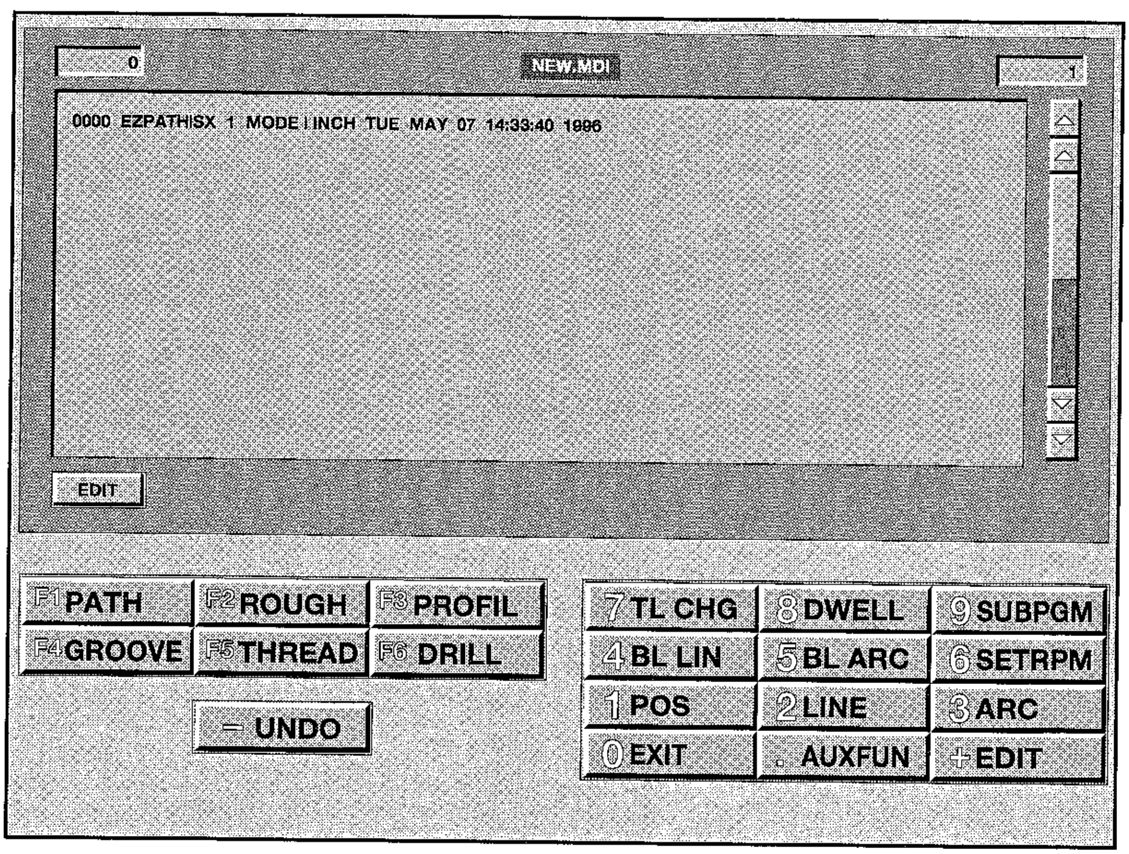
This command cuts the long taper.

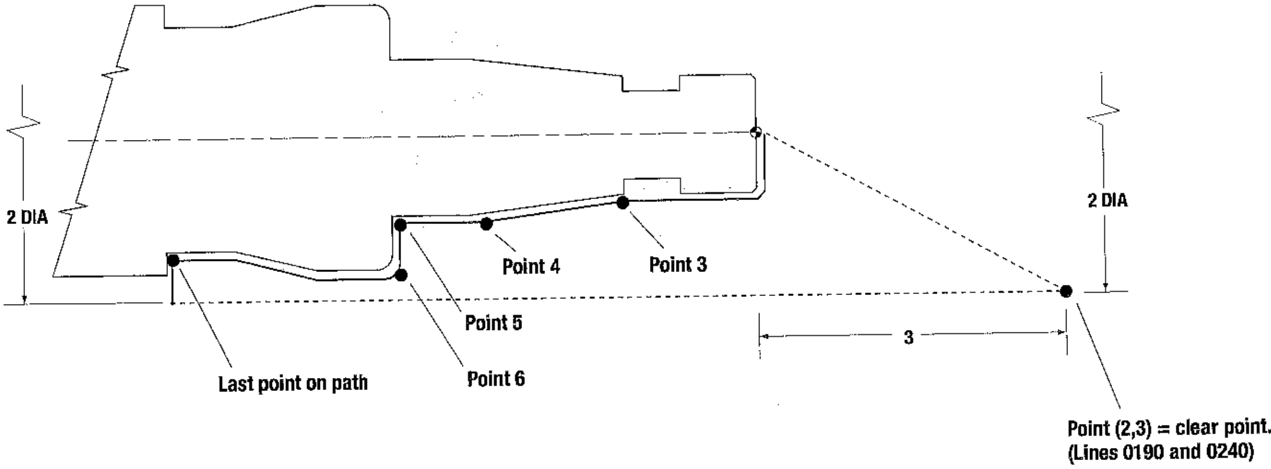
Fill in the dialog box with the coordinates of Point 4.





Above, part blueprint reference; ***below,*** MDI command screen





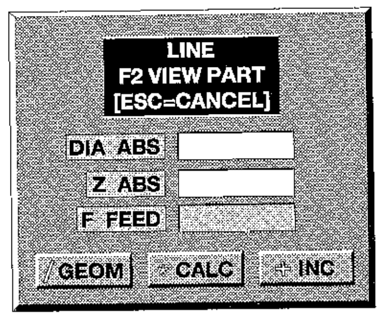
2 LINE

0060 LINE ABS X1.0000 Z-2.3750 F0.0100

MDI command: Line

This command cuts the flat surface after the taper.

Fill in the dialog box with the coordinates of Point 5.



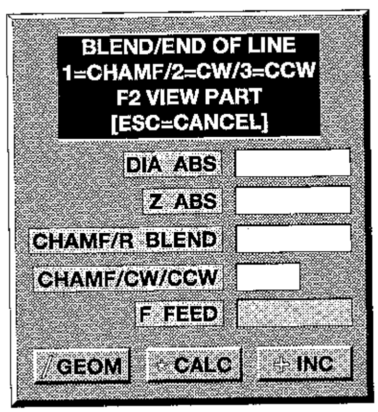
4 BL LIN

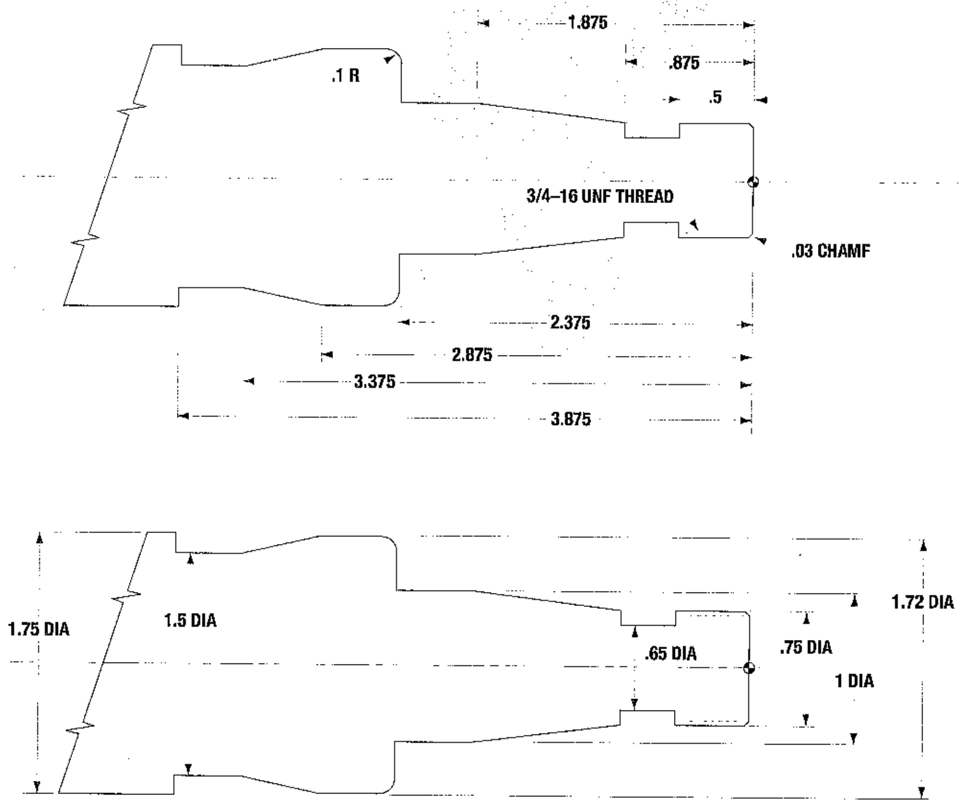
0070 BLENDILN ABS X1.7200 Z-2.3750 R0.1000 CW F0.0100

MDI command: Blend Line

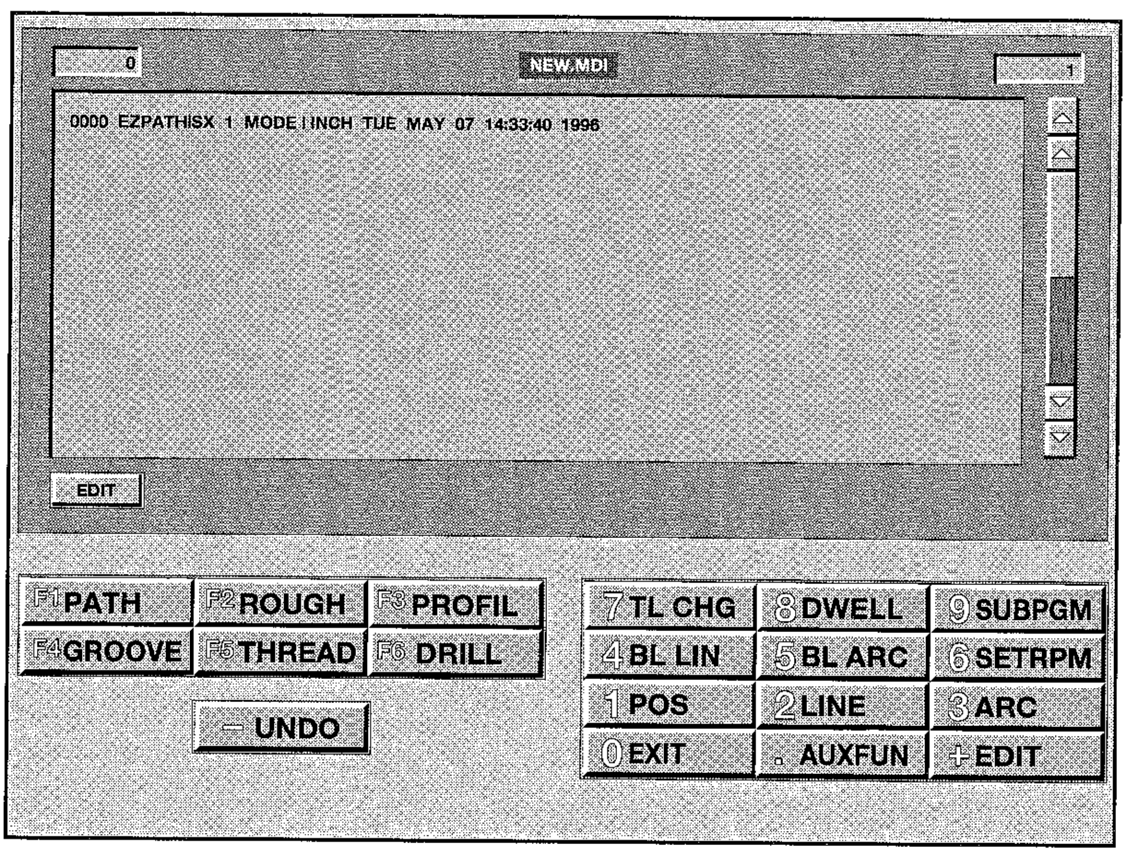
We just used the Blend Line to cut a chamfer; here, it's used to cut the .1" radius, by specifying "2" (CW radius) in the final data-entry box. Otherwise, it works just like the chamfer command: the tool begins cutting toward point (1.72,–2.375), but before it makes it to that point, it begins cutting the .1" radius such that the radius will blend perfectly with the line programmed in the next statement.

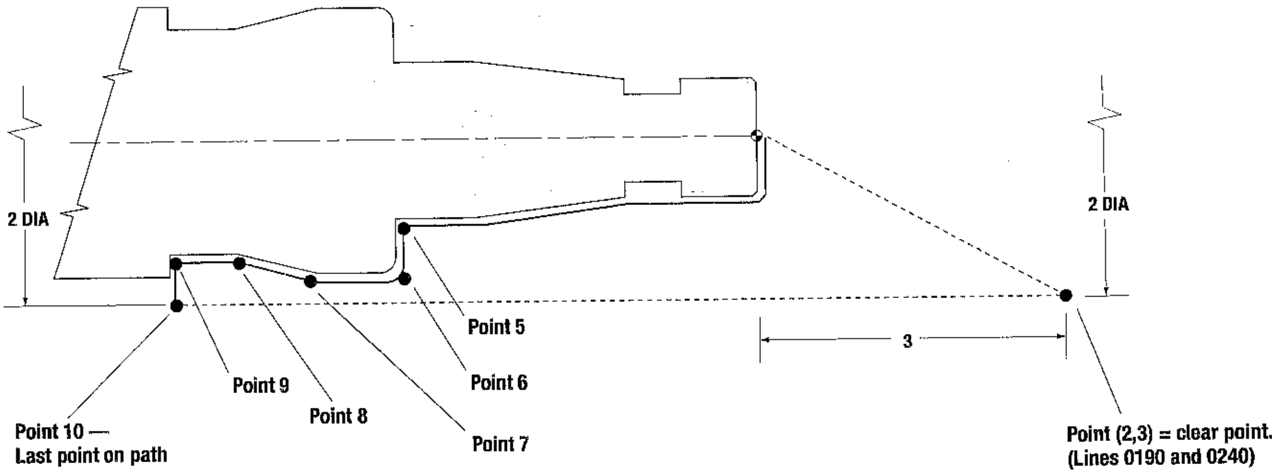
Fill in the dialog box with the coordinates of Point 6 and the radius specifications.





Above, part blueprint reference; ***below,*** MDI command screen





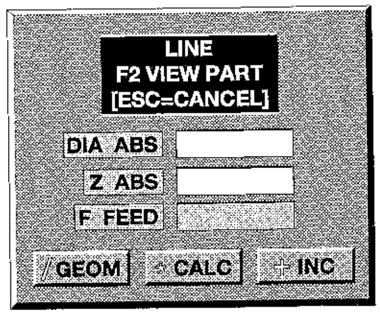
2 LINE

0080 LINE ABS X1.7200 Z-2.8750 F0.0100

MDI command: Line

This command cuts the outermost diameter of the part, blended with the radius of the previous command.

Fill in the dialog box with the coordinates of Point 7.



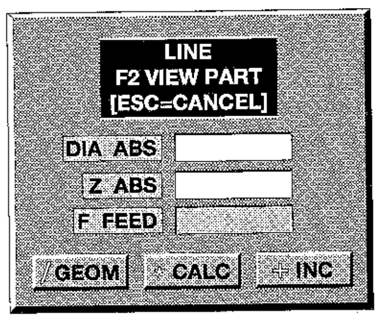
2 LINE

0090 LINE ABS X1.5000 Z-3.3750 F0.0100

MDI command: Line

This command cuts the taper on the back of the part. The ability to include a cut-back like this as part of a path is one of the EZPATH's innovative features.

Fill in the dialog box with the coordinates of Point 8.



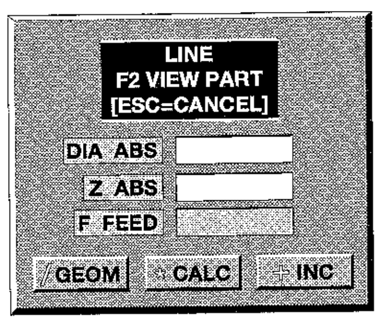
2 LINE

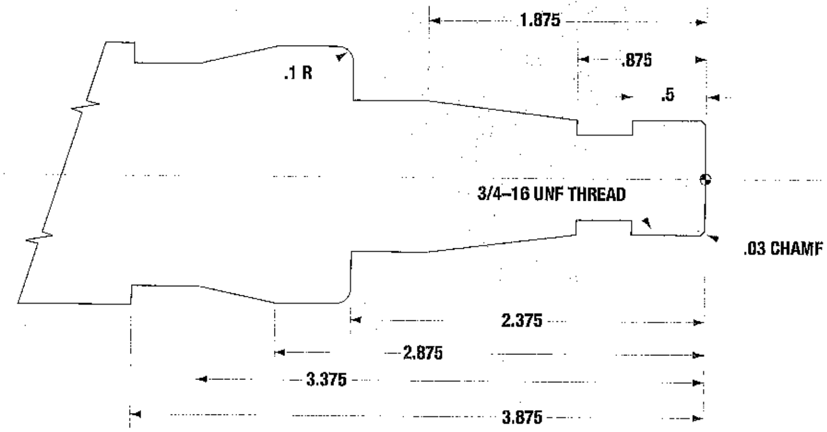
0100 LINE ABS X1.5000 Z-3.8750 F0.0100

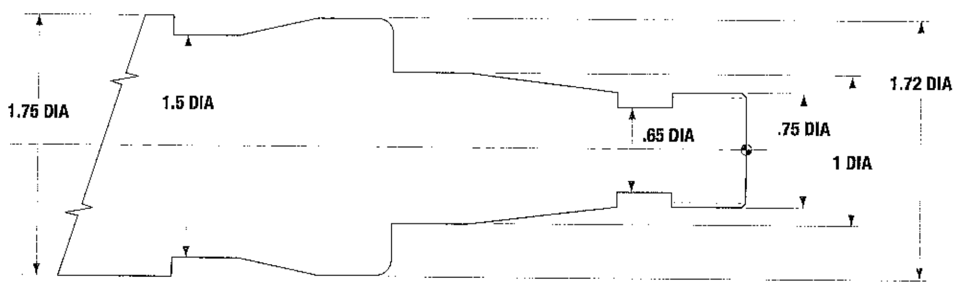
MDI command: Line

This command cuts the flat surface after the taper to the end of the part.

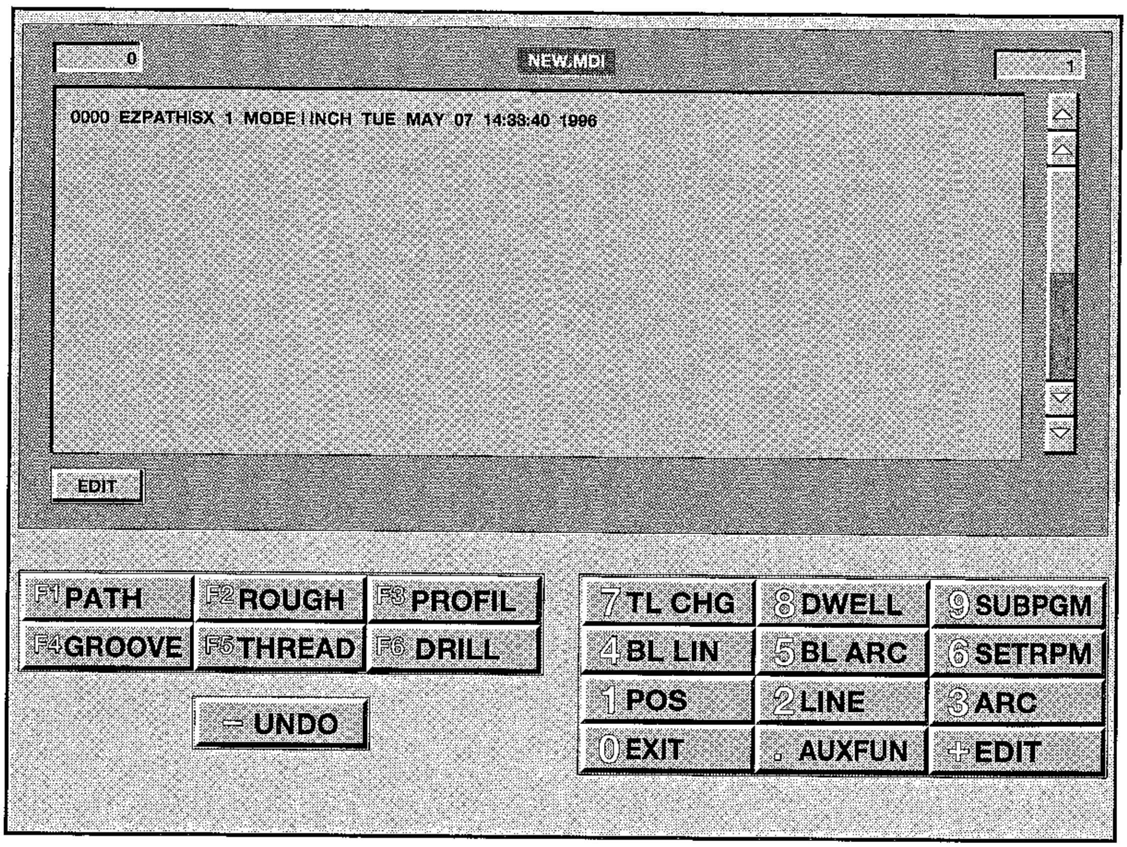
Fill in the dialog box with the coordinates of Point 9.

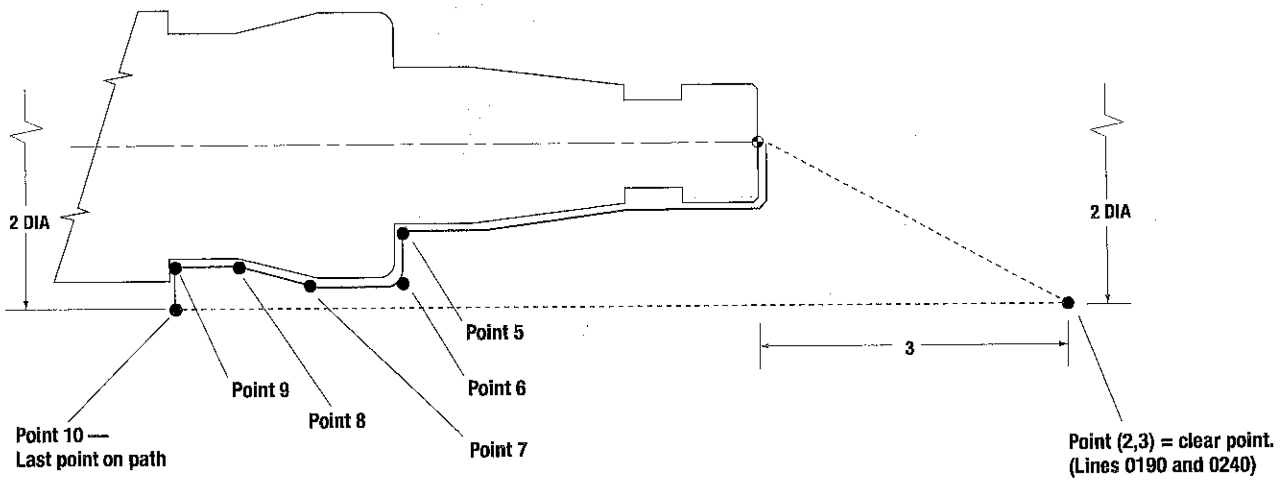






Above, part blueprint reference; ***below,*** MDI command screen





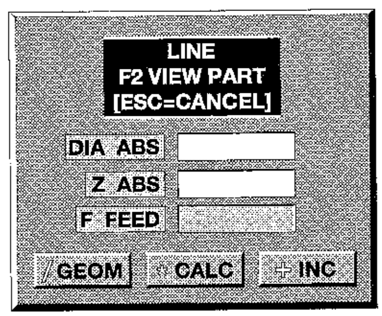
2 LINE

0110 LINE ABS X2.0000 Z-3.8750 F0.0100

MDI command: Line

This command cuts to the last point on the path: a straight line along the X-axis, moving the tool away from the part to a safe clearance distance.

Fill in the dialog box with the coordinates of Point 10.



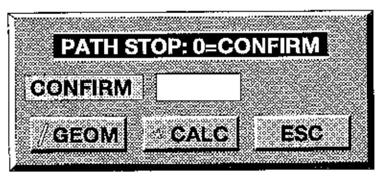
6 PTHSTP

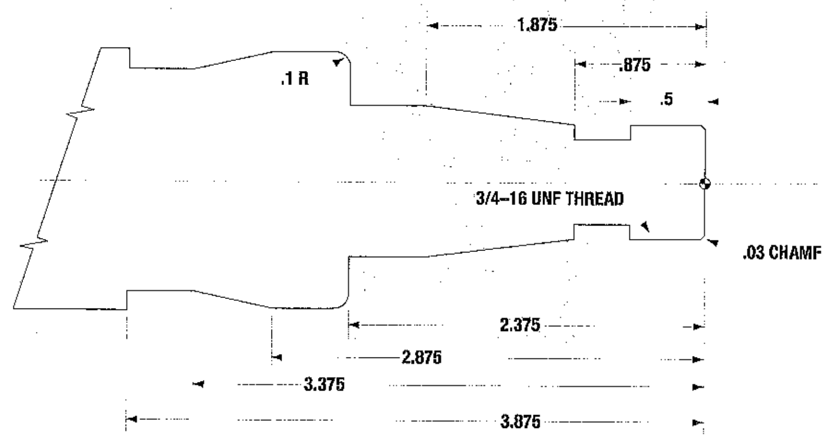
0120 PATHSTOP

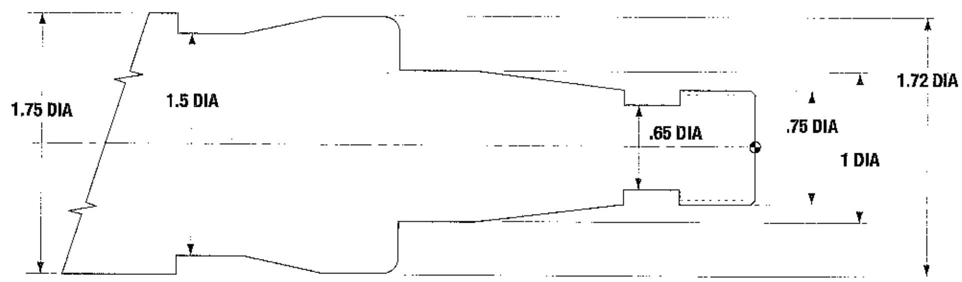
MDI command: Pathstop

Once the tool has been programmed to a safe point, enter the Path Stop command to terminate the Path definition. The command will require that you enter a "0" to confirm.

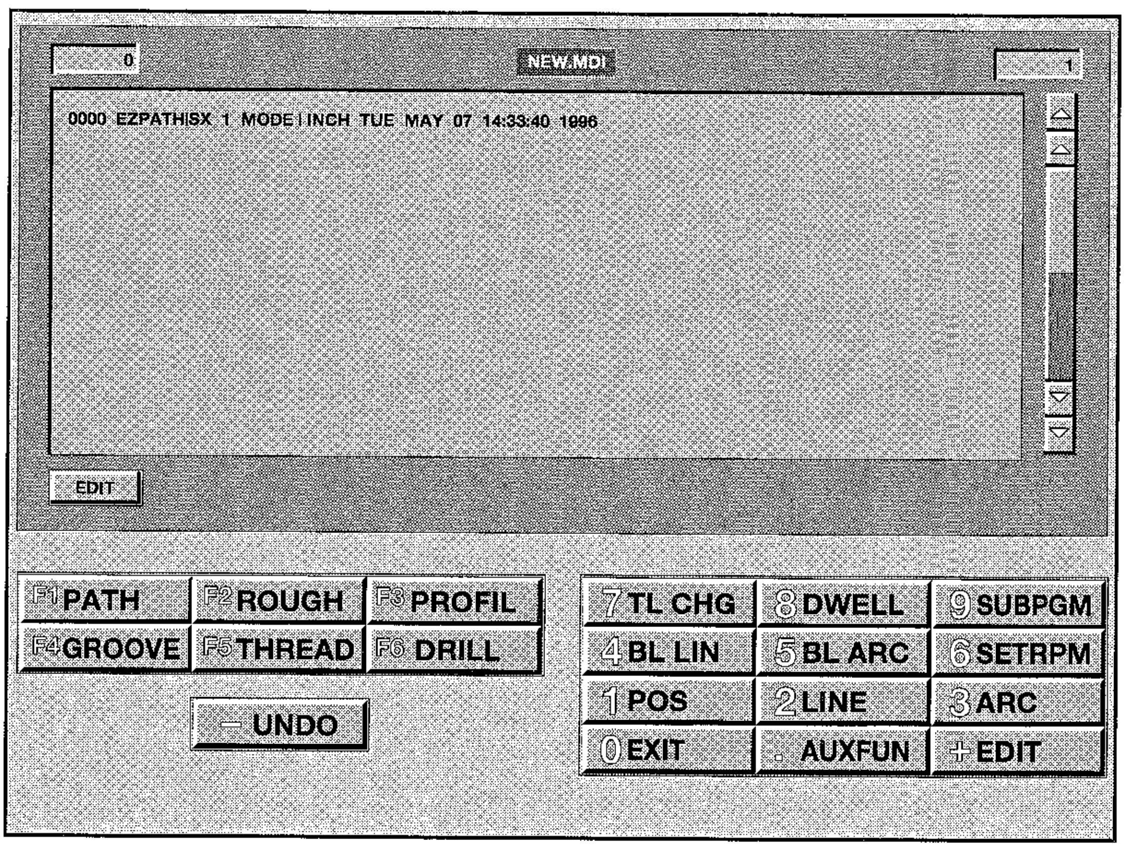
Fill in the dialog box: confirm end of path.

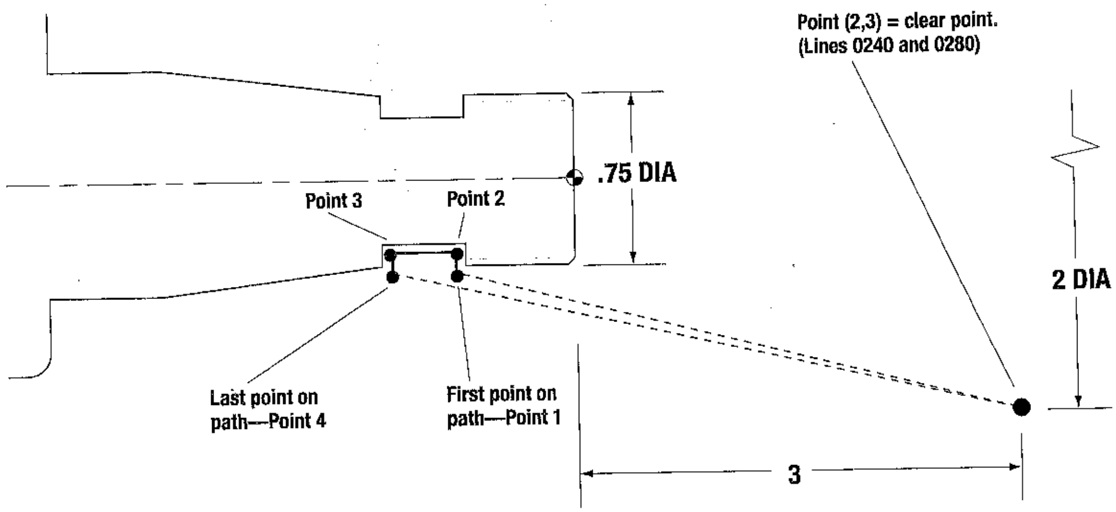






Above, part blueprint reference; ***below,*** MDI command screen





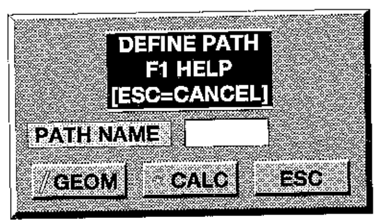
Creating the second path The heavy line shows the shape defined by the second path. You need to make sure that the tool has a clear route from the last point *before* the path to the first point *on* the path. *(Note that the distances in this diagram are somewhat exaggerated.)*

F1 PATH

0130 STARTPATH 2

MDI command: Define Path

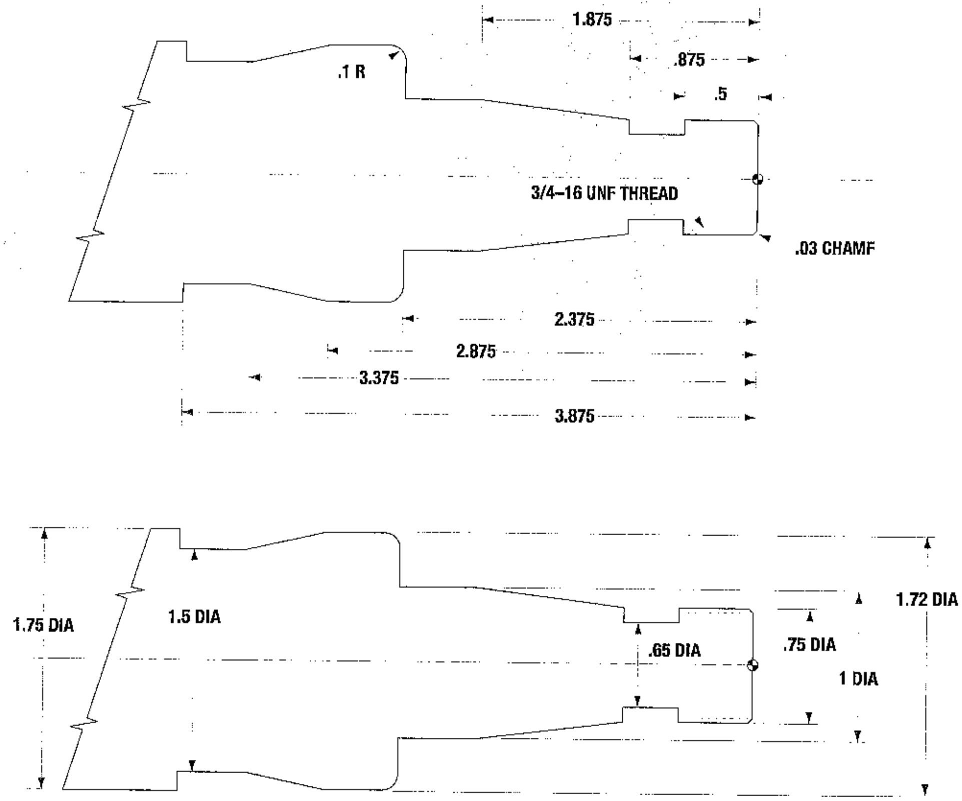
We may as well define the second path here; since path definition instructions don't involve the tool moving at all, we can define the path anywhere we choose (so long as it is before the canned cycle which calls it); however, it makes the program easier to understand if we do the path definitions near the beginning.



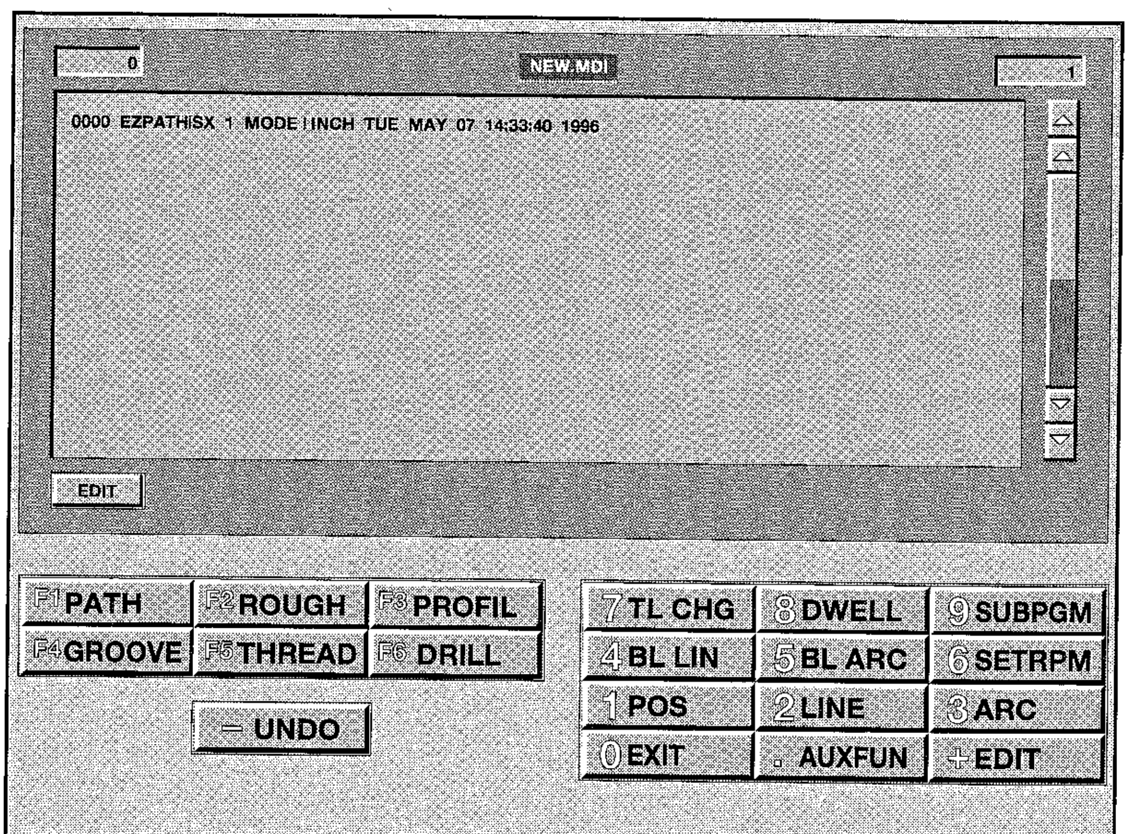
As mentioned earlier, this second path just cuts the inner groove in back of the first thread. Why do we need a second path just for this cut? Because while most of the part will be cut with a turning tool, for this groove we want to use a grooving tool. There is no place in a path definition to stop for a tool change, so all the elements in any single path definition must be capable of being cut with the same tool; no matter how simple the shape, if you want to change to a different tool, you must define a different path for the second (and any subsequent) tool. Also, each canned cycle uses a different type of tool motion to cut the path shape, as illustrated in the diagrams of sections 5.3.2–5.3.4, and for this particular piece of the part shape, the tool motion produced by the Groove command is more appropriate than Rough or Profile.

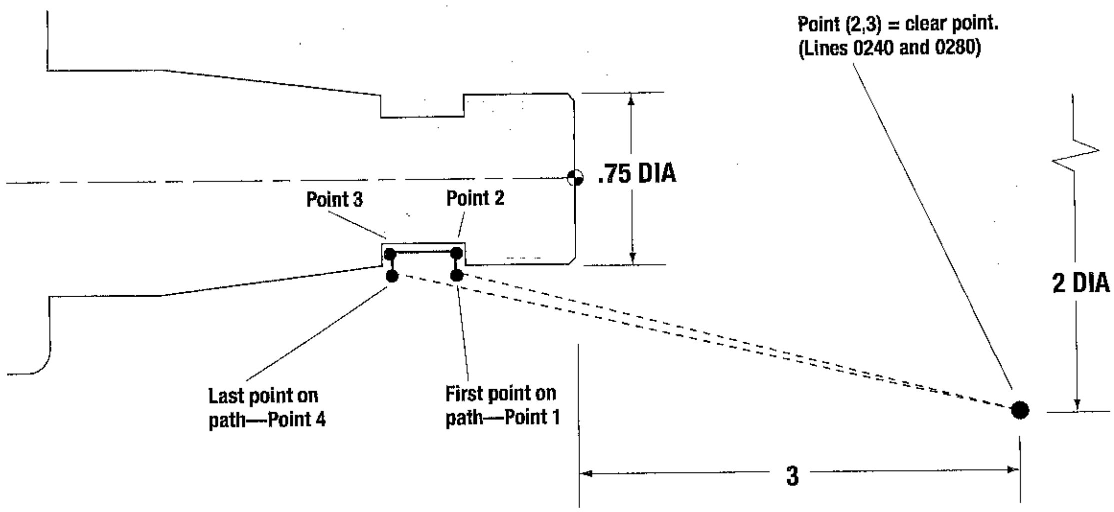
The above drawing illustrates the path for the groove.

Fill in the dialog box with the number of the path.



Above, part blueprint reference; ***below,*** MDI command screen



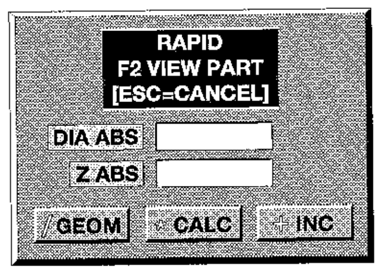


0140 RAPID ABS X0.7500 Z-0.5000

MDI command: Position

As mentioned earlier, the first command in any Path definition must be a rapid positioning move to the first point of the path.

Fill in the dialog box with the coordinates of Point 1.

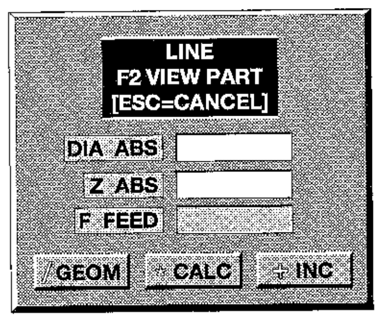


0150 LINE ABS X0.6500 Z-0.5000 F0.0100

MDI command: Line

This command causes the tool to feed into the part, beginning the groove.

Fill in the dialog box with the coordinates of Point 2.

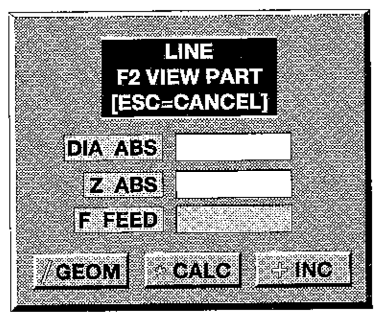


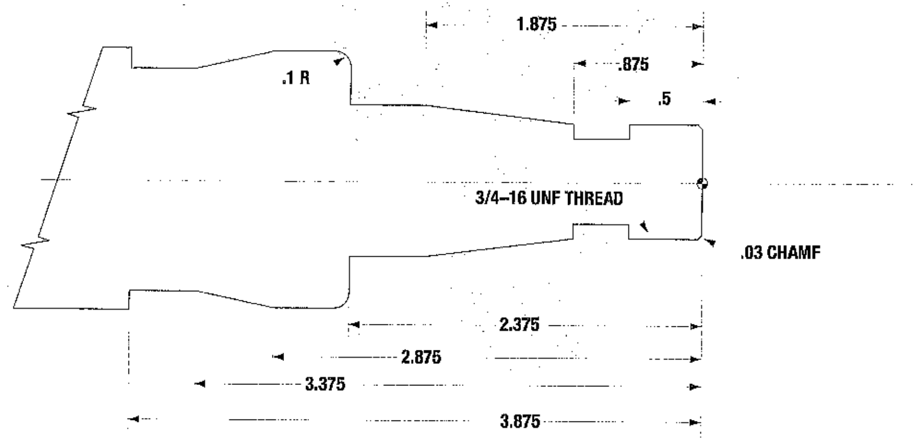
0160 LINE ABS X0.6500 Z-0.8750 F0.0100

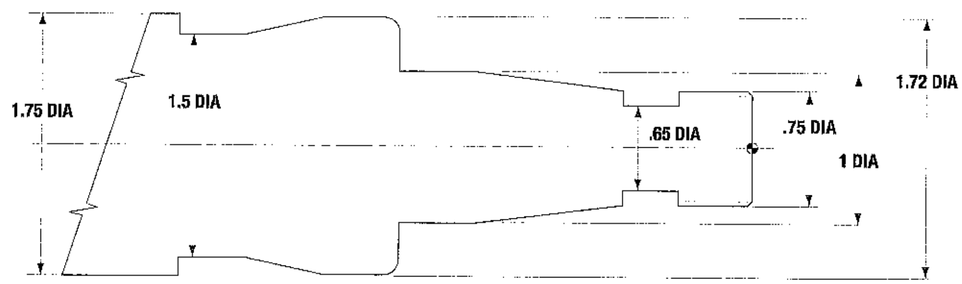
MDI command: Line

This command crossfeeds along the bottom of the groove.

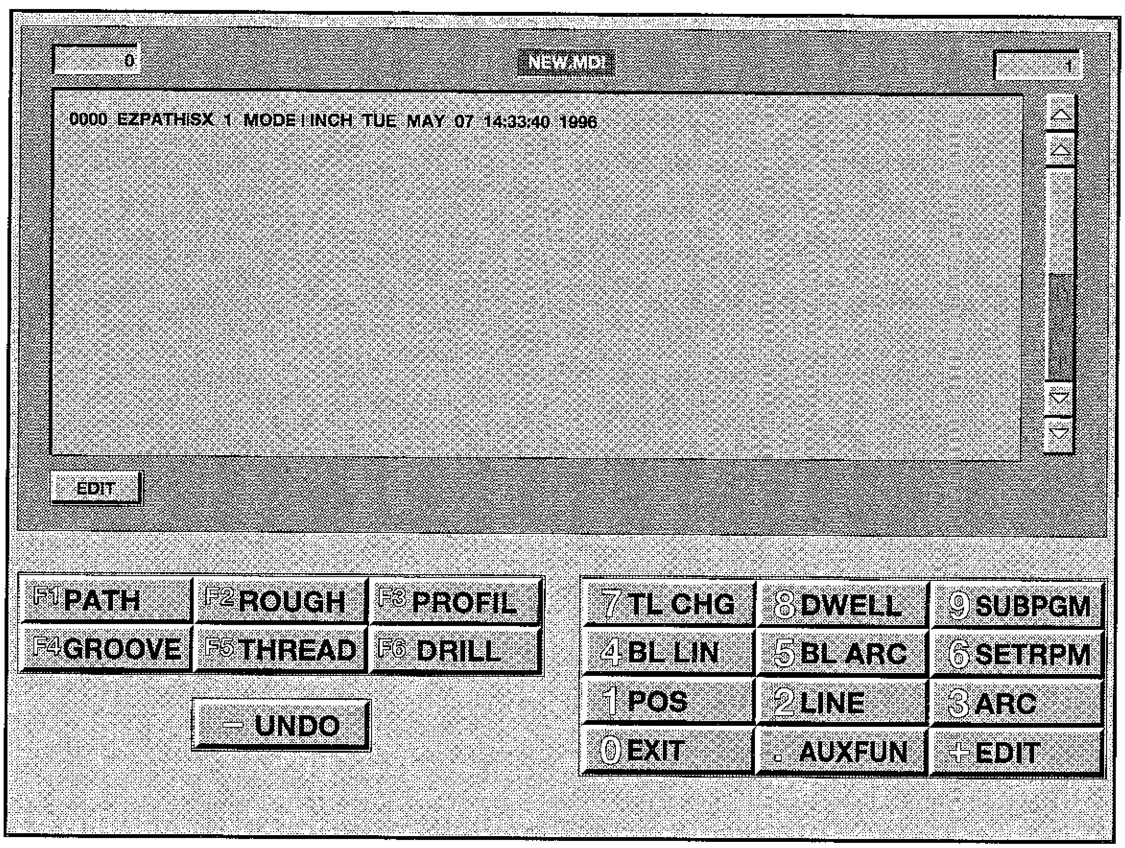
Fill in the dialog box with the coordinates of Point 3.

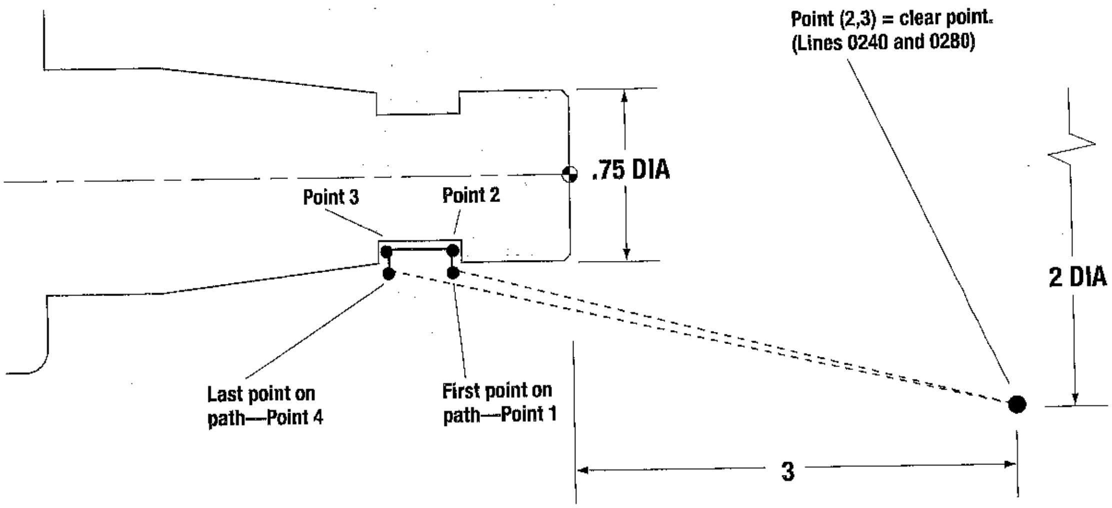






Above, part blueprint reference; ***below,*** MDI command screen





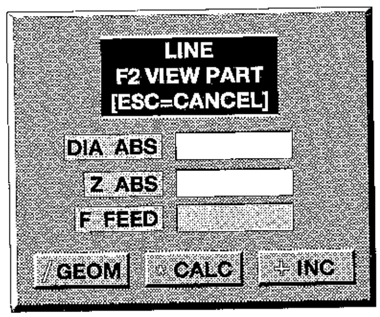
2 LINE

0170 LINE ABS X0.7500 Z-0.8750 F0.0100

MDI command: Line

This command cuts the back of the groove and pulls away from the part.

Fill in the dialog box with the coordinates of Point 4.



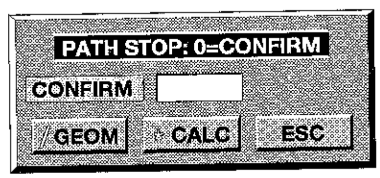
6 PTHSTP

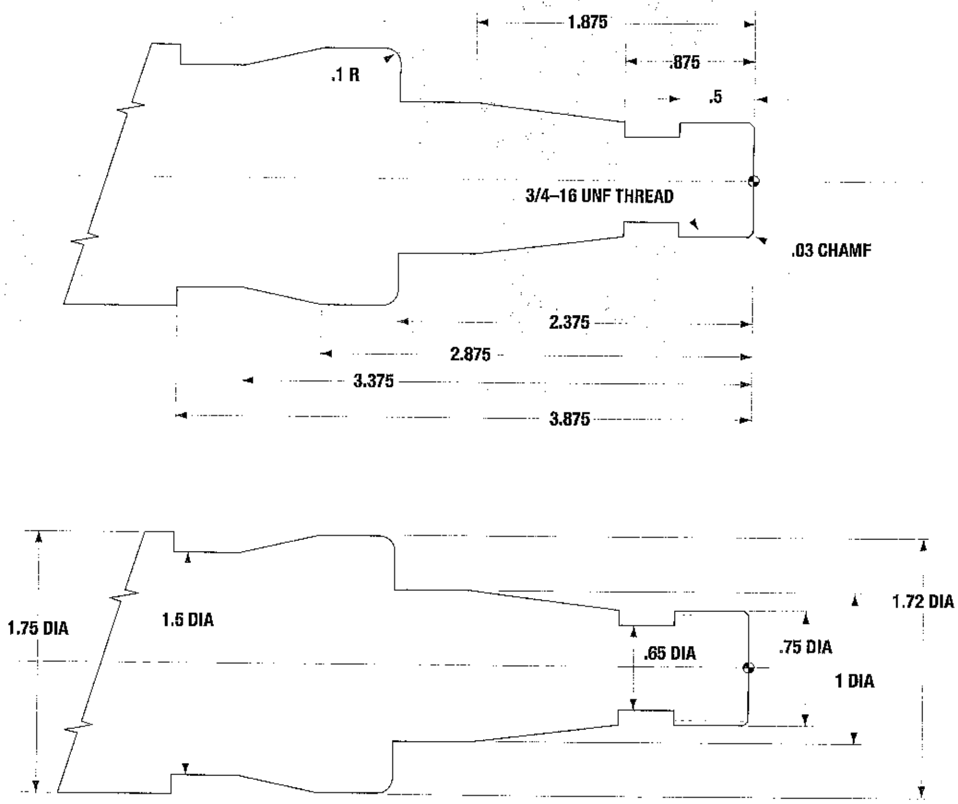
0180 PATHSTOP

MDI command: Pathstop

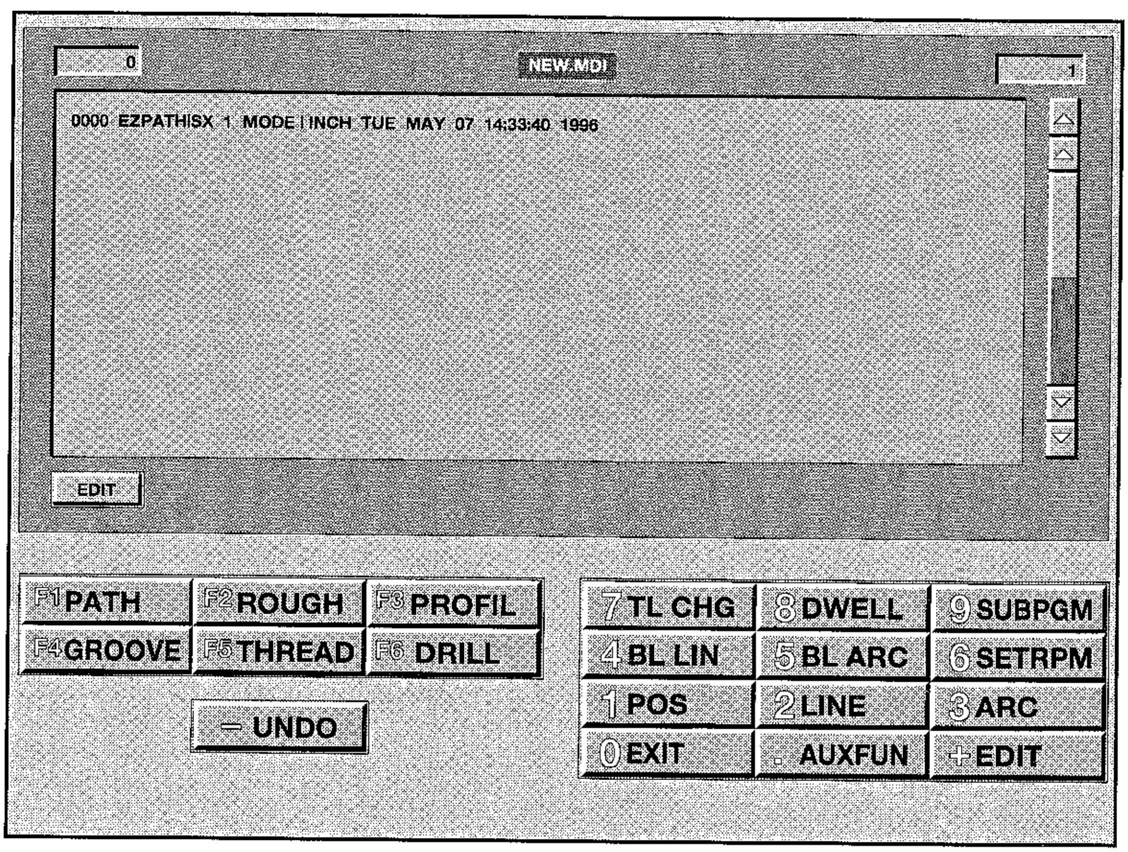
This command ends the second path definition. Now, we can begin the main body of the program.

Fill in the dialog box: confirm end of path.



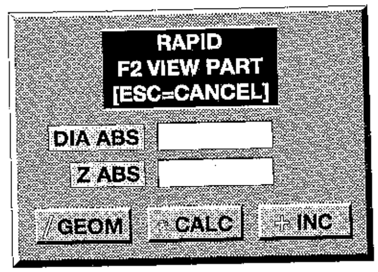


Above, part blueprint reference; ***below,*** MDI command screen



1 POS

0190 RAPID ABS X2.0000 Z3.0000



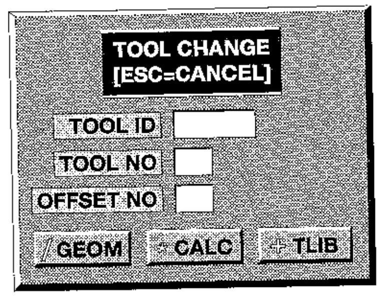
MDI command: Position

Forget for a moment all of the positioning and cutting commands in the path definitions. This is the first instruction that actually causes the machine to *do* something visible—it moves the tool carriage to the point (2,3), a safe clearance point so that the tool turret can rotate to the proper position. This point is also a safe location from which the tool can move to the starting point of each path.

Fill in the dialog box with the coordinates of the clear point.

7 TL CHG

0200 TLCHG I11 T01 01



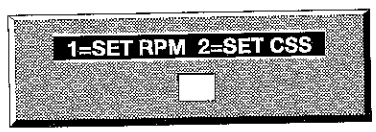
MDI command: Tool Change

When the EZPATH reaches this instruction, the program pauses and prompts the operator to insert tool number 1—the OD turning tool— in the tool holder. The tool ID is what EZPATH uses to reference the tool dimensions in the tool library. The proper tool offset must be set prior to executing the program. The tool IDs, numbers, and offsets used here are examples only; be sure to use values appropriate for your shop.

Fill in the dialog box with the number, ID, and offset of the OD turning tool.

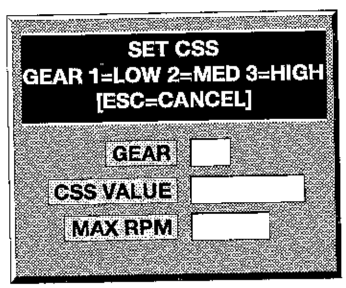
6 SETRPM

0210 SETCSS G3 C675.00 S3200

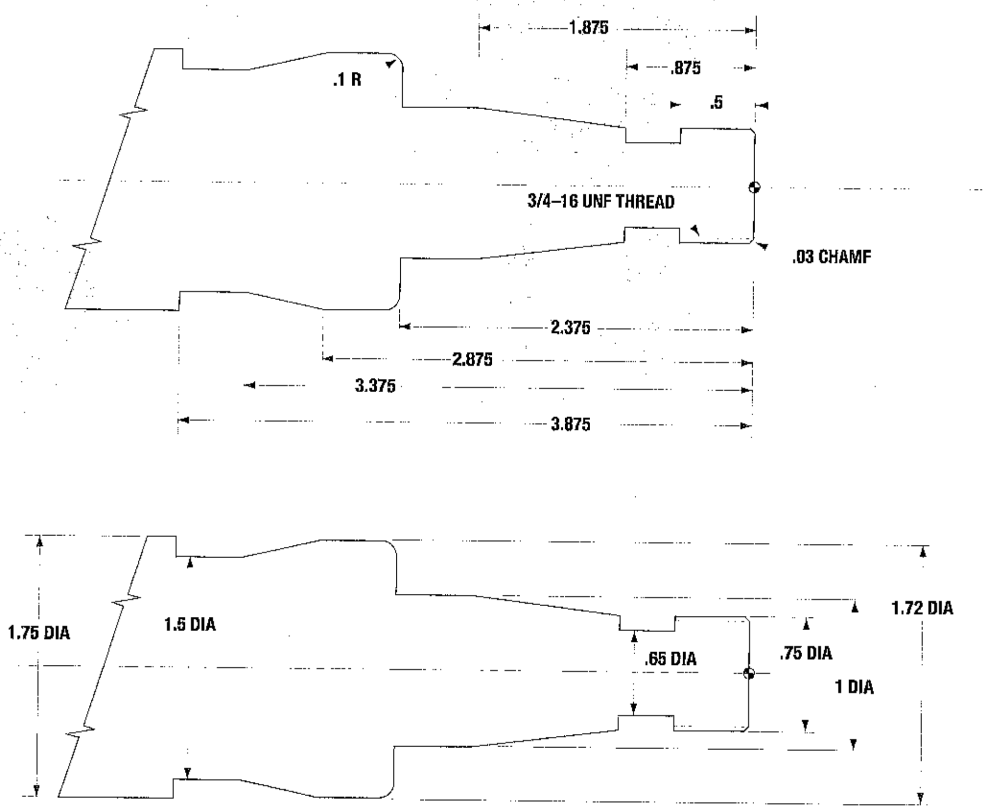


MDI command: Set RPM

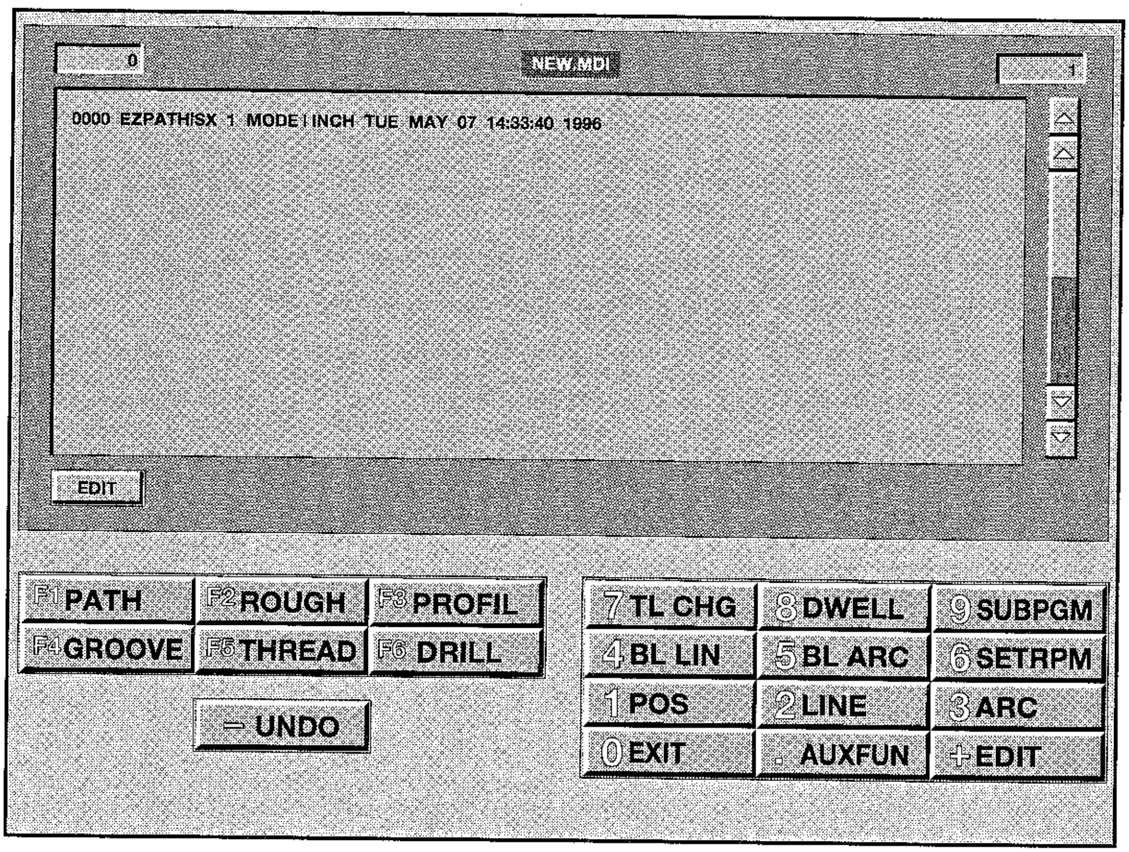
After pressing the 6 SETRPM key, press 2 to select the Set CSS command (instead of Set RPM). The EZPATH spindle has three gears; enter "3" to select the higher one (35–3,600 rpm), which will produce spindle speeds appropriate for the part-diameter/surface-speed combinations in this part. Note that you will still need to place the machine in the proper gear manually, using the lever on the front of the machine. The MAX RPM value shown here may not be appropriate for all chucks; verify an appropriate value with a supervisor *before* running this program.



Fill in the dialog box with the proper constant surface speed parameters.



Above, part blueprint reference; ***below,*** MDI command screen

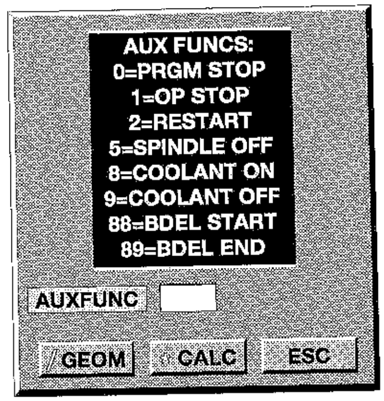


0220 AUXFUN M8

MDI command: Auxiliary Functions

Enter this function to turn on the spindle (clockwise direction), and at the same time turn on the coolant. This single command is equivalent to an M3 code followed by an M8.

Fill in the dialog box with the proper code.

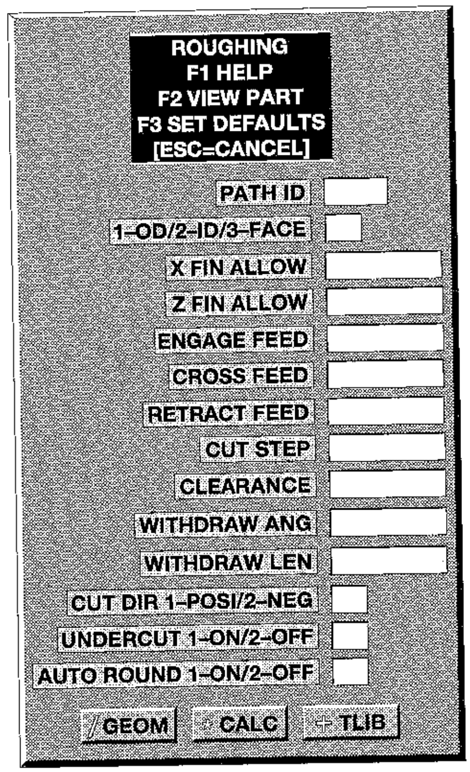


0230 ROUGH 1 I1 X0.0000 Z0.0000 F0.0100
0.0100 0.0100 S0.1000 C0.1250 W45.0000 W0.0500 D2
U1 A1

MDI command: Rough

The Rough command will now rough out the shape defined in Path 1. Note that the tool will move to the first point in the path definition (plus the clearance distance specified here) at the rapid traverse rate, in a straight line (vector)—not axis-by-axis. You must ensure that the route from the current tool location (in this case, the point [2,3]) to the first point in the path is clear of obstructions. See section 6.3.2 of the Programming manual for a detailed illustration of the tool motion created by this command and descriptions of each of the data elements it requires.

Fill in the boxes with the Roughing specifications.

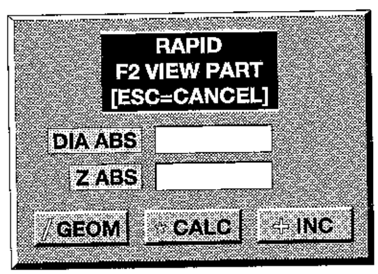


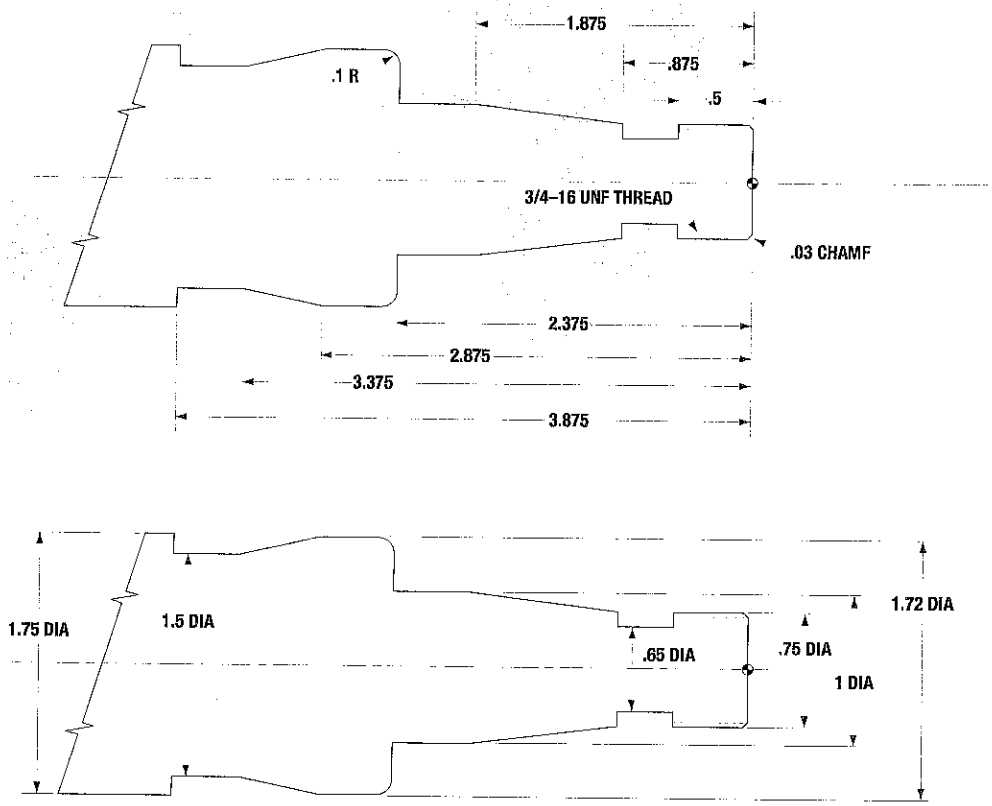
0240 RAPID ABS X2.0000 Z3.0000

MDI command: Position

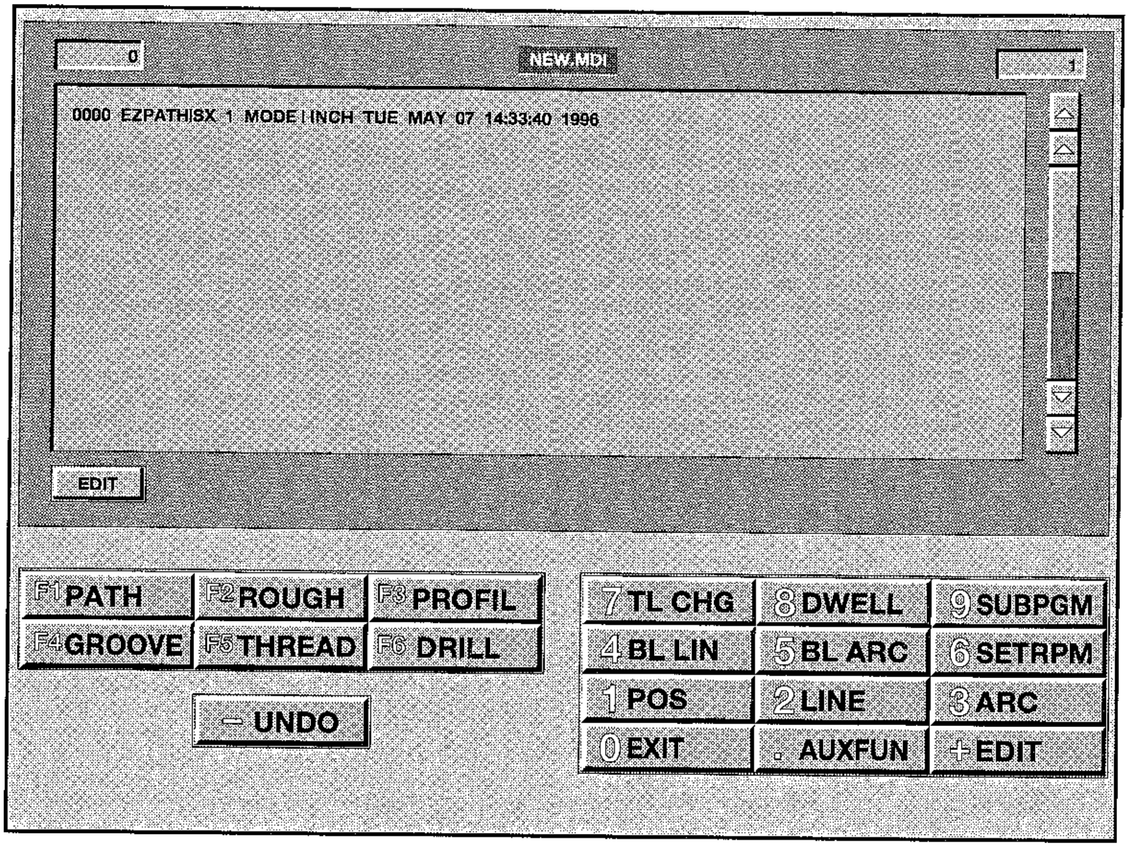
After the Roughing operation is complete, this Position command moves the tool from the last point defined in Path 1 (2,-3.875) to a suitable location for the next tool change and from where it can begin the Groove cycle.

Fill in the dialog box with the coordinates of the clear point.





Above, part blueprint reference; ***below,*** MDI command screen

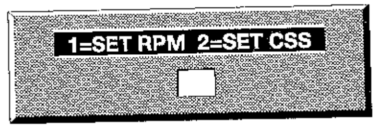


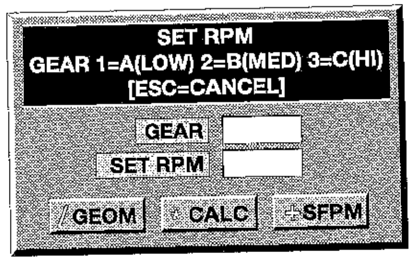
0250 SETRPM G3 S850

MDI command: Set RPM

After selecting the Set RPM command, enter "1" to select rpm mode instead of constant surface speed; then, enter the gear and spindle speed to be used for the upcoming Groove operation.

Fill in the dialog box with the proper gear and spindle speed.



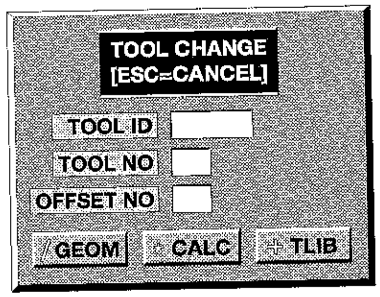


0260 TLCHG I93 T02 02

MDI command: Tool Change

The program will now stop and prompt the operator to install tool number 2. This is the OD grooving tool that will be used to in the upcoming Groove command. The tool IDs, numbers, and offsets used here are examples only; be sure to use values appropriate for your shop.

Fill in the dialog box with the number, ID, and offset of the OD grooving tool.

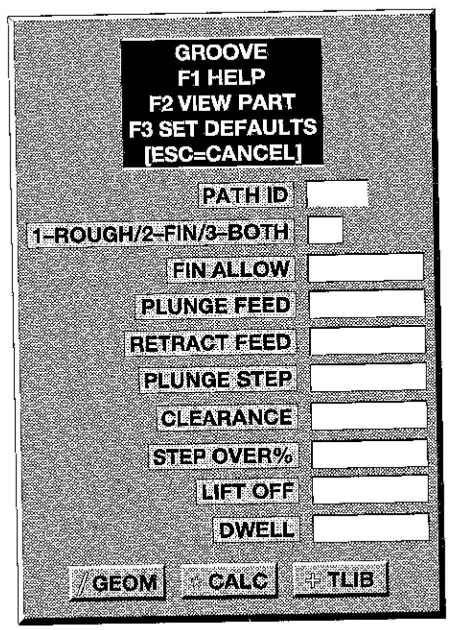


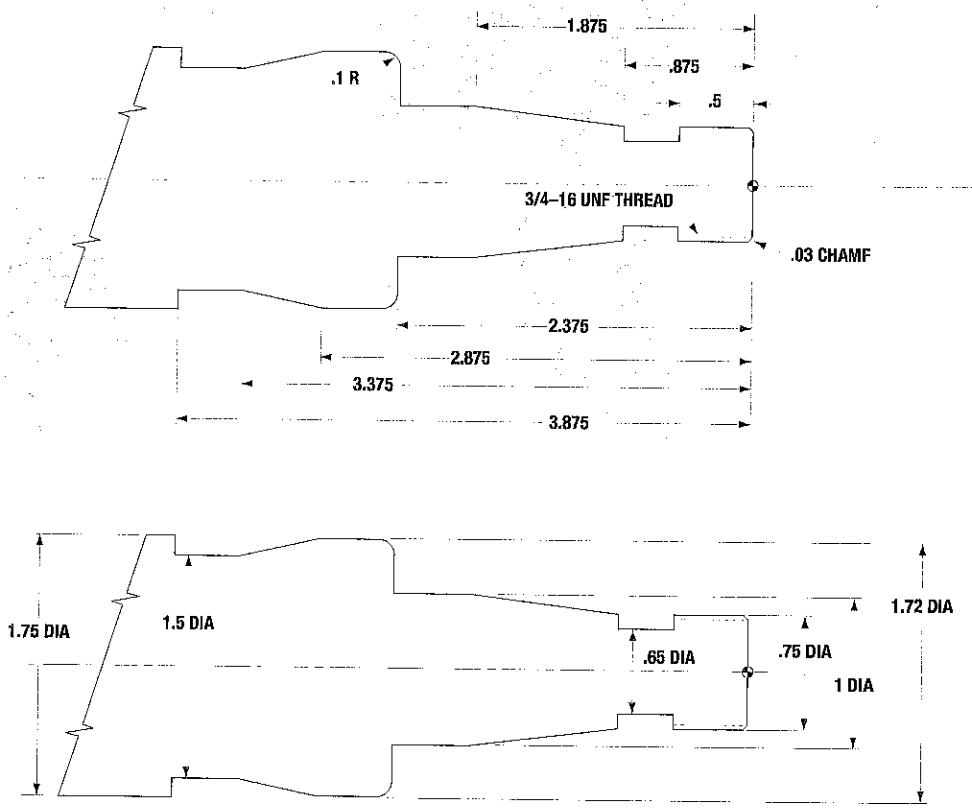
0270 GROOVE 2 3 A0.0100 F0.0100 R0.0100 P0.0000 C0.1000 O80.0000 L0.0500 D1.0000

MDI command: Groove

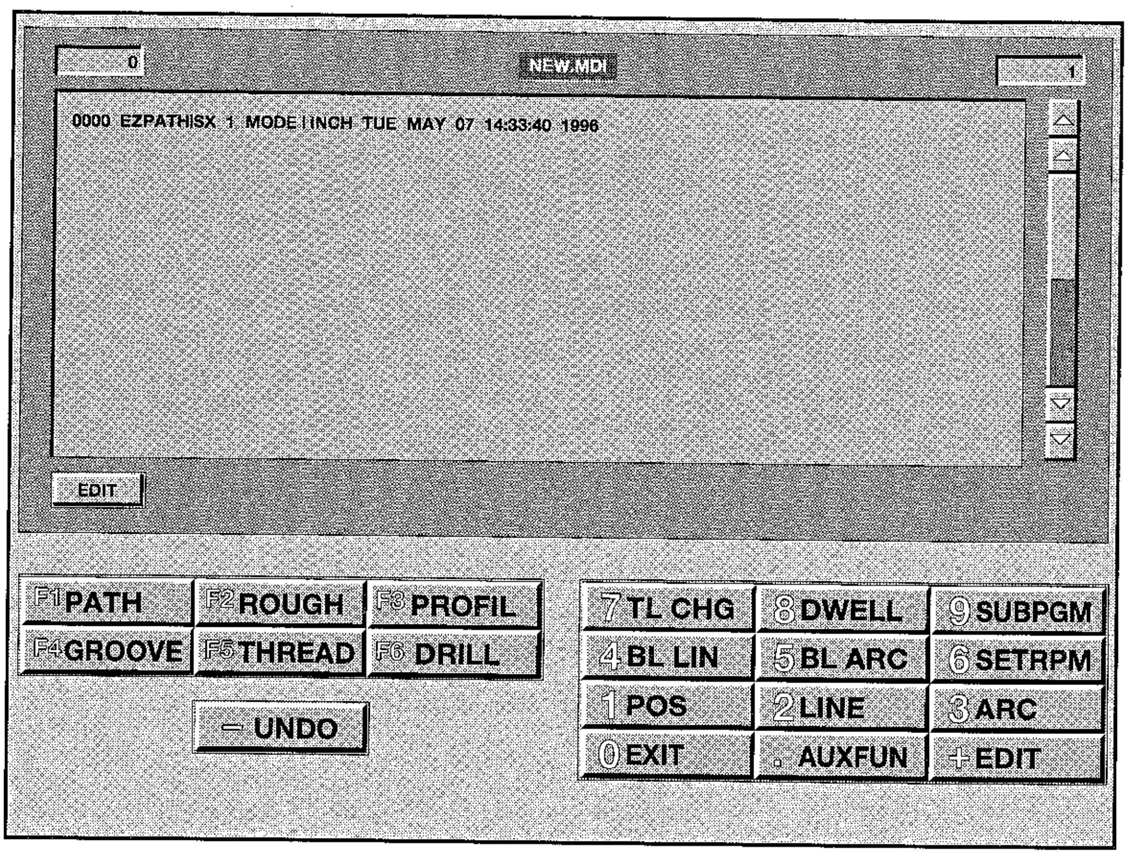
Now that the proper tool is in place and the spindle speed programmed, you can use the Groove command to cut the groove. With the Groove command, distinct roughing and finishing cycles can be programmed within one command; for this operation, you will want to perform both roughing and finishing passes. Section 6.3.4 of the Programming manual illustrates the tool motion produced by each type of operation and describes the meaning of each required data element.

Fill in the dialog box with the information necessary to complete the Groove cycle.





Above, part blueprint reference; ***below,*** MDI command screen



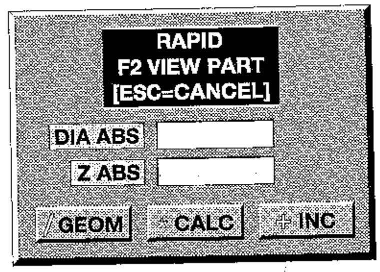
1 POS

0280 RAPID ABS X2.0000 Z3.0000

MDI command: Position

Once again, after the canned cycle is completed, position the tool at a safe location for a tool change—this time, in preparation for cutting the threads.

Fill in the dialog box with the coordinates of the clear point.



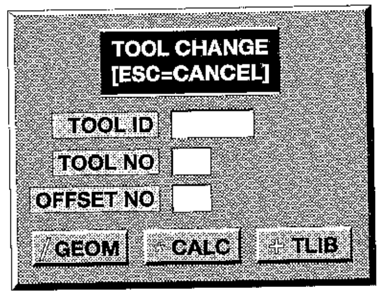
7 TL CHG

0290 TLCHG I60 T03 03

MDI command: Tool Change

This command prompts the operator to install number 3, the 60° threading tool that will be used in the upcoming Thread command. Remember, the tool IDs, numbers, and offsets used here are examples only; be sure to use values appropriate for your shop.

Fill in the dialog box with the number, ID, and offset of the threading tool.



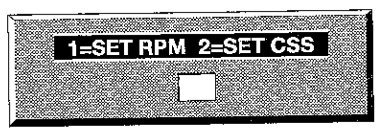
6 SETRPM

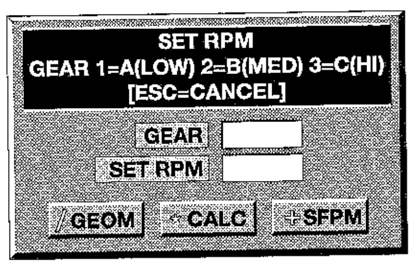
0300 SETRPM G3 S530

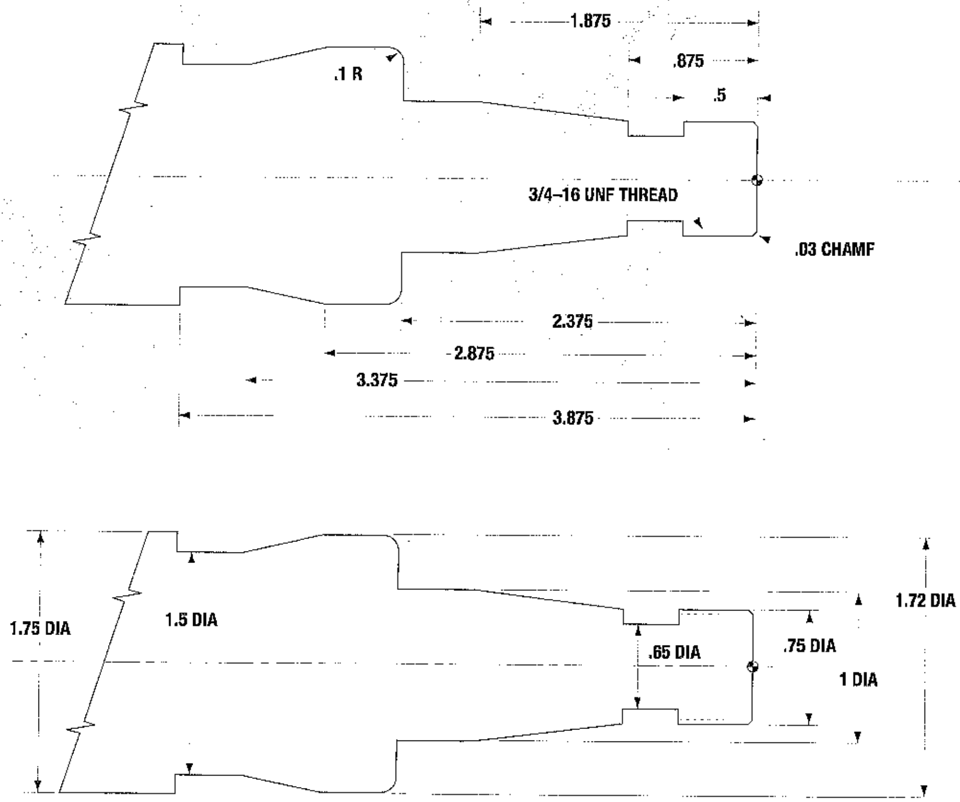
MDI command: Set RPM

Before the Thread command, set the spindle speed to an appropriate value.

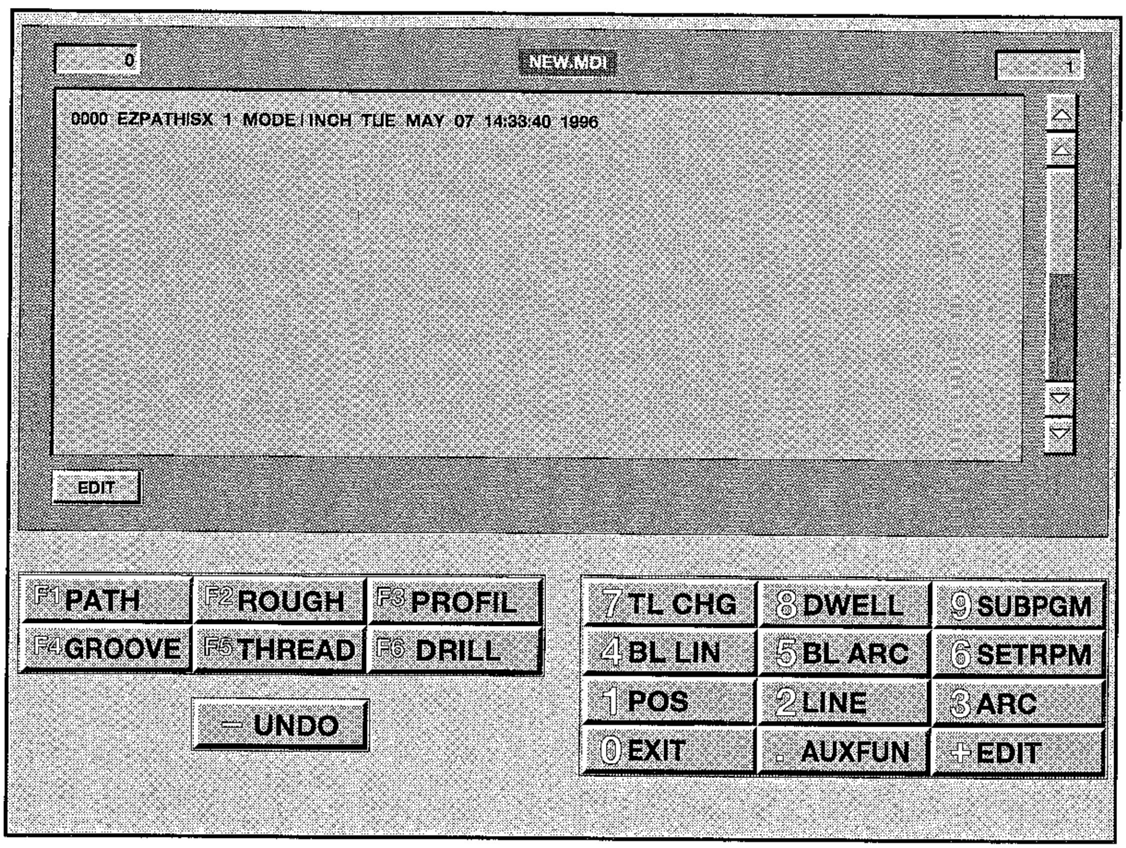
Fill in the dialog box with the proper spindle speed.







Above, part blueprint reference; ***below,*** MDI command screen



F5 THREAD

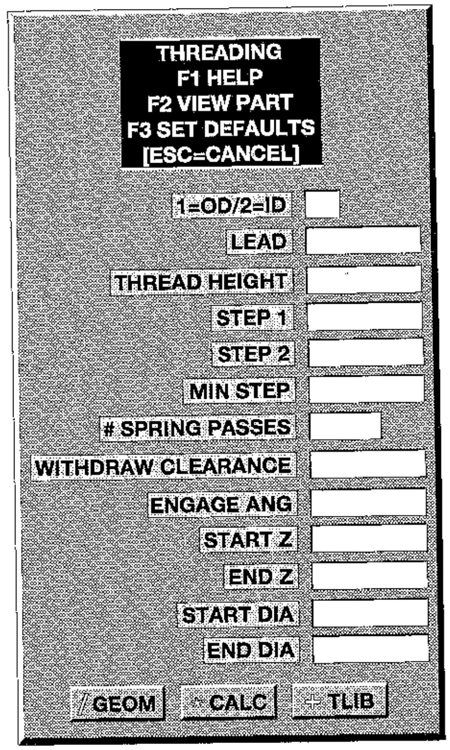
0310 THREAD 1 L0.0625 H0.0472 S0.0050 0.0030 0.0020 #2 C0.1000 Z0.1500 -0.5500 D0.7500 0.7500 29.0000

MDI command: Thread

Use the Thread command to cut the front set of threads. Note that when you are entering the data above, a thread height will be calculated for you automatically, based on a 60° UN thread; if your thread will be different, just type over this value with any number you wish. Section 6.3.5 of the manual describes each of the data elements needed by the Threading cycle.

This is the last cutting operation; the part is now complete.

Fill in the dialog box with the information necessary to complete the Thread cycle.



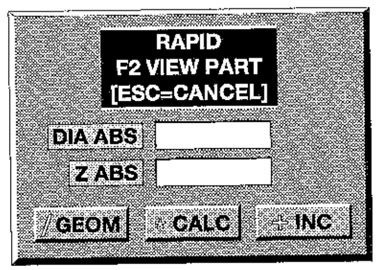
1 POS

0320 RAPID ABS X2.0000 Z3.0000

MDI command: Position

After the threads are cut, this positioning move pulls the tool far enough away from the part to remove the part from the chuck and/or inspect it, and to otherwise prepare the machine for the next operation.

Fill in the dialog box with the coordinates of the clear point.



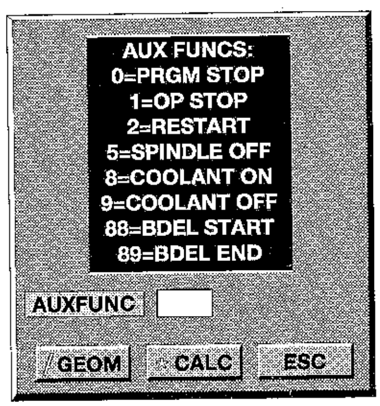
AUXFUN

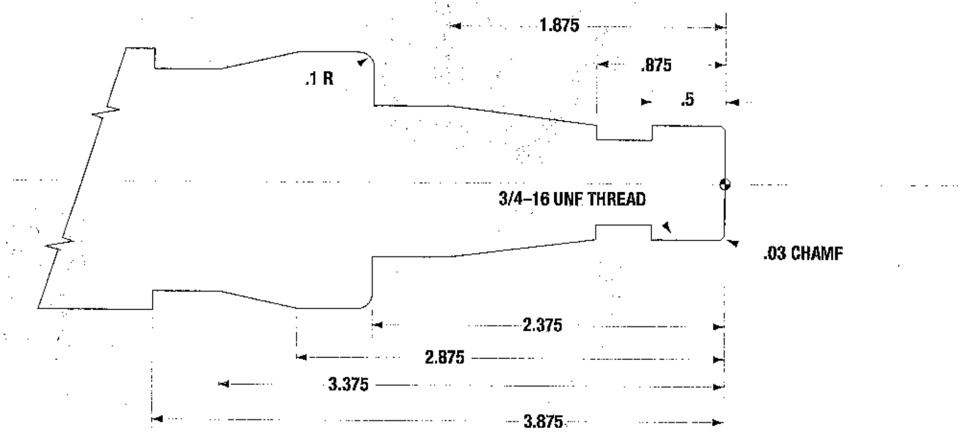
0330 AUXFUN M2

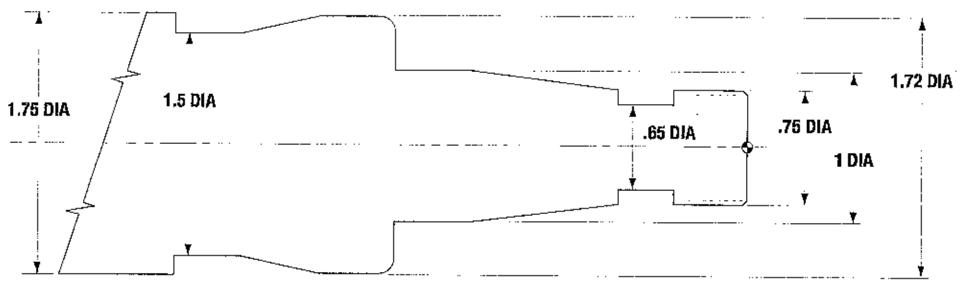
MDI command: Auxiliary Function

This code stops the program, prompts the operator to turn off the spindle, and resets the program to the beginning, so that it can be immediately run again with having to use the Run: Reset command.

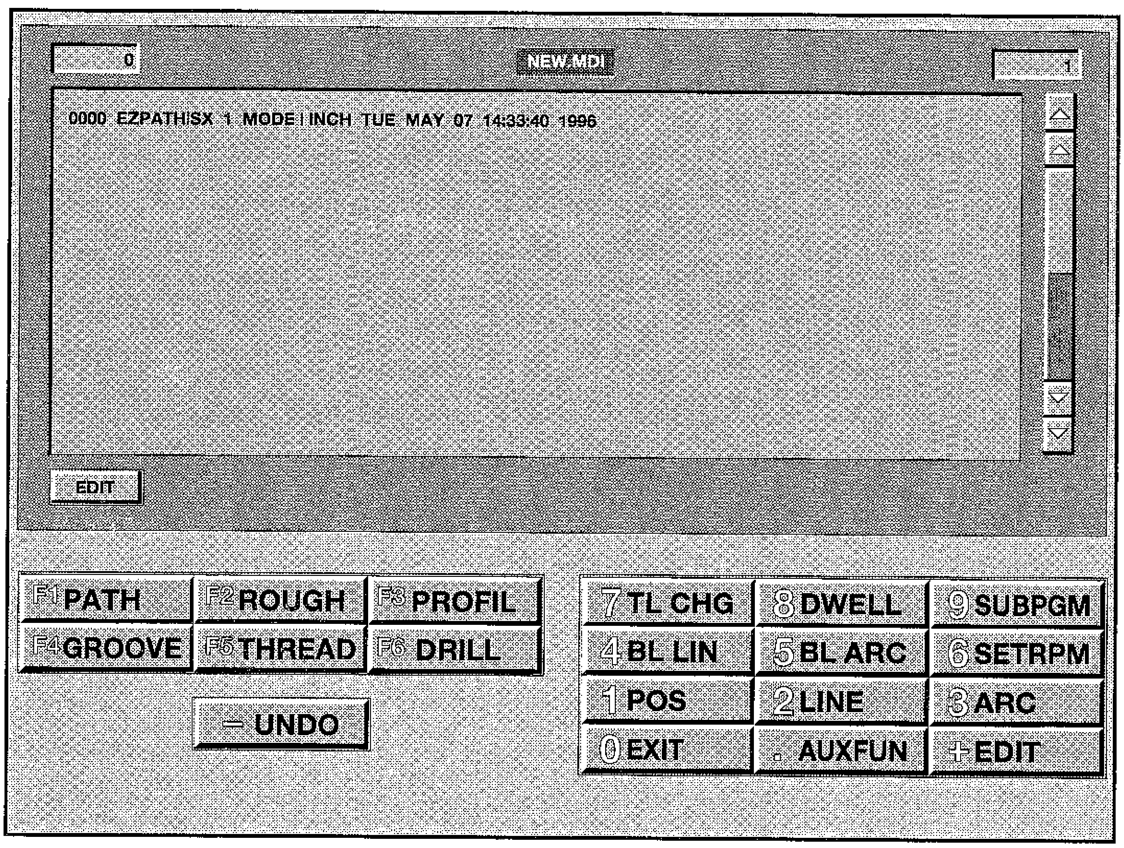
Fill in the dialog box with the proper code.







Above, part blueprint reference; ***below,*** MDI command screen



0 EXIT

Your program is now complete. When you select the Exit command, you will have the opportunity to save your program and assign it a name/number. Section 6.1.3 of the manual describes in detail how to save your program and your options for exiting MDI mode.

For your convenience, the program is listed again below:

```
0000 EZPATHISX 1 MODEIINCH SUN JUL 07 20:08:05 1996
0010 STARTPATH 1
0020 RAPID ABS X0.0000 Z0.0000
0030 CHAMFER ABS X0.7500 Z0.0000 P0.0300 P0.0300 F0.0100
0040 LINE ABS X0.7500 Z-0.8750 F0.0100
0050 LINE ABS X1.0000 Z-1.8750 F0.0100
0060 LINE ABS X1.0000 Z-2.3750 F0.0100
0070 BLENDILN ABS X1.7200 Z-2.3750 R0.1000 CW F0.0100
0080 LINE ABS X1.7200 Z-2.8750 F0.0100
0090 LINE ABS X1.5000 Z-3.3750 F0.0100
0100 LINE ABS X1.5000 Z-3.8750 F0.0100
0110 LINE ABS X2.0000 Z-3.8750 F0.0100
0120 PATHSTOP
0130 STARTPATH 2
0140 RAPID ABS X0.7500 Z-0.5000
0150 LINE ABS X0.6500 Z-0.5000 F0.0100
0160 LINE ABS X0.6500 Z-0.8750 F0.0100
0170 LINE ABS X0.7500 Z-0.8750 F0.0100
0180 PATHSTOP
0190 RAPID ABS X2.0000 Z3.0000
0200 TLCHG I11 T01 01
0210 SETCSS G3 C675.00 S3200
0220 AUXFUN M8
0230 ROUGH 1 I1 X0.0100 Z0.0100 F0.0100 0.0100 0.0100 S0.1000 C0.1250W45.0000W0.0500D2U1A1
0240 RAPID ABS X2.0000 Z3.0000
0250 SETRPM G3 S850
0260 TLCHG I93 T02 02
0270 GROOVE 2 3 A0.0030 F0.0100 R0.0100 P0.0000 C0.0130 O80.0000 L0.0500 D1.0000
0280 RAPID ABS X2.0000 Z3.0000
0290 TLCHG I60 T03 03
0300 SETRPM G3 S530
0310 THREAD 1 L0.062500 H0.0383 S0.0050 0.0030 0.0020 #2 C0.1000 Z0.1500 -0.5500D0.7500 0.7500 29.0000
0320 RAPID ABS X2.0000 Z3.0000
0330 AUXFUN M2
```