EZTRAK®

Programming and Operating Manual

Control Operating Software
Version 6.00/5.78

Code No. 1104-2909

March 2001

Bridgeport®

BRIDGEPORT MACHINES, INC.

Accredited by
the Dutch Council
for Certification

ISO 9001
Registered
KPMG
Quality Registrar
The Quality Advantage

IMPORTANT

SAFETY INFORMATION

To prevent bodily injury, you should observe the following basic safety precautions when installing, operating or servicing your Bridgeport Machines, Inc. machine.

1. Follow all instructions in the manual.
2. Wear approved industrial safety glasses and safety shoes.
3. **DO NOT** wear gloves, long sleeves, long hair, rings, watches, jewelry or other items that could become caught in moving parts.
4. Keep all parts of your body away from moving parts (belts, cutters, gears, etc.).
5. Use proper point of operation safeguarding.

These and other safety precautions are discussed in the American National Standards Institute Standard entitled ***Safety Requirements for the Construction, Care, and Use of Drilling, Milling, and Boring Machines*** (ANSI B11.8-1983).

This publication is available from:

The American National Standards Institute
1430 Broadway
New York, NY 10018

Safeguarding for protection at the point of operation can only be designed and constructed when the parameters of the particular operation have been determined. As a result, ANSI B11.8-1983, Section 5.1, states that ***"it shall be the responsibility of the employer to provide, and ensure the use of, a guard, guarding device, awareness barrier, awareness device, or shield..."***

To assist machine users in designing point of operation safeguarding for their specific machine applications the Occupational Safety and Health Administration has published a booklet entitled ***Concepts and Techniques of Machine Safeguarding*** (O.S.H.A. Publication Number 3067).

This publication is available from:

The Publication Office - O.S.H.A.
U.S. Department of Labor
200 Constitution Avenue, NW
Washington, D.C. 20210

The general purpose point of operation shield provided with this machine may not be appropriate and cannot be utilized for all possible applications of the machine. Use additional or alternate safeguarding where this shield is not appropriate or cannot be utilized. Note that for purposes of display, the shield has been removed in certain illustrations in this manual.

BRIDGEPORT SAFETY LIST
ALL COUNTRIES

1. **DON'T** run your machine until you have read and understood the Bridgeport Operator and Maintenance manuals.
2. **DON'T** run your machine until you have read and understood all machine and control key signs.
3. **DON'T** run your machine for the first time without a qualified instructor. **ASK** your supervisor for help when you need it.
4. **PROTECT** your eyes. Wear safety glasses with side shields at all times.
5. **DON'T** get caught in moving parts. Remove watches, rings, jewelry, neck-ties, and loose-fitting clothes.
6. **PROTECT** your head. Wear a safety helmet when working near overhead hazards.
7. **KEEP** your hair away from moving parts.
8. **PROTECT** your feet. Always wear safety shoes with steel toes and oil-resistant soles.
9. Gloves are easily caught in moving parts. **TAKE THEM OFF** before you turn on the machine.
10. **REMOVE** all loose items (wrenches, chuck keys, rags, etc.) from machine before starting. Loose objects can become flying particles.
11. **NEVER** operate a machine tool after taking strong medication, using non-prescription drugs or consuming alcoholic beverages.
12. **SAFEGUARD** the cutting zone ("point of operation"). Use standard, general purpose safeguard where possible. Use special safeguards when required.
13. Protect your hands. **STOP** the spindle completely **BEFORE** changing tools.
14. Protect your hands. **STOP** the spindle completely **BEFORE** you load or unload a workpiece.
15. Protect your hands. **STOP** the spindle completely **BEFORE** you clear away chips or oil. Use brush or chip scraper. **NEVER** use you hands.
16. Protect you hands. **STOP** the spindle completely **BEFORE** you adjust the workpiece, fixture, or coolant nozzle.
17. Protect your hands. **STOP** the spindle completely **BEFORE** you take measurements.
18. Protect your hands. **STOP** the spindle completely **BEFORE** you open safeguard or covers.
19. Never reach around a safeguard.
20. Protect your hands. **STOP** the machine **BEFORE** you change or adjust belts, pulleys or gears.
21. **PROTECT** your hands. Keep hands and arms clear of spindle start switch when changing tools.
22. **PROTECT** your eyes and the machine. **Never use a compressed air hose to remove chips.**

23. **KEEP** work area well-lighted. Ask for additional light if needed.
24. **DON'T** slip. Keep your work area clean and dry. Remove chips, oil and obstacles.
25. **NEVER** lean on your machine. Stand away when the machine is running.
26. **DON'T** get trapped. Avoid pinch points caused by motion of table and head.
27. **PREVENT** objects from flying loose. Securely clamp and locate workpiece. Use stop blocks where necessary. **KEEP** clamps clear of cutter path.
28. **PREVENT** cutter breakage. Use correct table feed and spindle speed for the job. Reduce feed and speed if you notice unusual noise or vibration.
29. **PREVENT** cutter breakage. Rotate spindle in clockwise direction for right-hand tools, counterclockwise for left-hand tools. Use the correct tool for the job.
30. **PREVENT** workpiece and cutter damage. Never start the machine when the cutter is in contact with the workpiece.
31. **KEEP** tools sharp. Dull and damaged tools break easily. Inspect tools and tool holders. Keep tool overhang short.
32. Keep rotating cranks and handwheels well-lubricated and maintained. Do not remove safety springs.
33. Certain materials, such as magnesium, are highly flammable in dust and chip form. See your supervisor before working with these materials.
34. **PREVENT** fire. Keep flammable liquids and materials away from work area and hot chips.
35. **PREVENT** machine table from moving unexpectedly. Disengage power feed when not being used (manual machines only).
36. **PREVENT** machine from moving unexpectedly. Always start machine in manual mode.

WARNING

ANY UNAUTHORIZED CHANGING OF CONTROL PARAMETERS IS NOT PERMITTED. BRIDGEPORT MACHINES WILL NOT ACCEPT ANY LIABILITY WHATSOEVER FOR THE ALTERATION OF ANY SET PARAMETERS TO THOSE PROGRAMMED AT INSTALLATION.

DANGER

Danger notices are used in the publication to emphasize that lethal electrical voltages are present which could cause serious personal injury or death.

WARNING

Warning notices are used in this publication to emphasize that hazardous mechanical conditions, voltages, currents, or temperatures exist in this equipment which could cause serious personal injury and/or damage to the equipment.

CAUTION

Caution notices are used where equipment might be damaged if care is not taken. In situations where inattention could cause either personal injury or damage to the equipment, a WARNING notice is used.

NOTE

Notes merely call attention to information that is especially significant in understanding and operating the equipment.

IMPORTANT NOTICE

DO NOT ATTEMPT DISASSEMBLY OR REMOVAL OF MAJOR COMPONENTS WITHOUT FIRST CONTACTING THE BRIDGEPORT MACHINES SERVICE DEPARTMENT FOR PROPER PROCEDURES.

This document is intended for the use of those who install, operate, and maintain the Bridgeport Milling Machine. Although reasonable care has been exercised in the preparation of this manual to make it complete and accurate, this manual does not purport to cover all conceivable problems or applications pertaining to this machine.

INTRODUCTION

EZTRAK® is an extension of Bridgeport's PCNC (PC-based Numerical Control) product line, providing operators the familiarity of a manual machine with the power of a CNC. The EZTRAK is designed for the first-time PCNC user, and is intended to bridge the gap between the handwheel-driven milling machine and the full-featured PCNC, with the ease of use of one, and the increased productivity of the other.

The EZTRAK is targeted at milling one of a kind parts or small job lots and gives the user the flexibility of several different modes of machining in one package:

- In the intelligent **3-AXIS DRO** (Digital Read Out) Mode, the machinist can use the advanced digital readout and the axes handwheels in the same way that the conventional milling machines in the shop are operated.
- In the **DO EVENT** Mode, a step by step conversational display prompts the operator for all the information required to easily machine arcs, rectangles, circles, slots and bolt circles, etc.
- In the **TEACH** Mode the operator can manually machine the first part and save the coordinates of each move to replay the operations for subsequent parts.
- In the **RUN** Mode, the EZTRAK® can automatically run the same exact part programs used by thousands of other BRIDGEPORT CNCs.

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

About This Manual

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

1) Chapter 1 - EZTRAK® Hardware & Software
2) Appendix A - Axes and Coordinates;
3) Chapter 2 - Starting Up the EZTRAK®;
4) Chapter 3 - Basic Operations;
5) Chapter 4 - Manual DRO & JOG Mode;
6) Chapter 5 - DO EVENT Mode; and
7) Chapter 6 - EZTRAK® Tutorial.

PLEASE NOTE: This book includes information for both 2 axis and 3 axis EZTRAKs. Differences in programming and operating requirements are duly noted in the man ual wherever they occur.

Table of Contents

Chapter 3
BASIC OPERATION

Chapter 4
MANUAL DRO & JOG OPERATION

Chapter 5
DO EVENT OPERATION

Chapter 6
EZTRAK® TUTORIAL

Chapter 7
MDI PROGRAMMING MODE

Chapter 8
EDIT MODE

Chapter 9
RUN MODE

Chapter 10
FILE UTILITIES

Chapter 11
TEACH MODE

Chapter 12
PREVIEW MODE

Chapter 1
HARDWARE & SOFTWARE

This section describes the basic components and controls of the EZTRAK® and should be reviewed carefully before cutting any part on the machine.

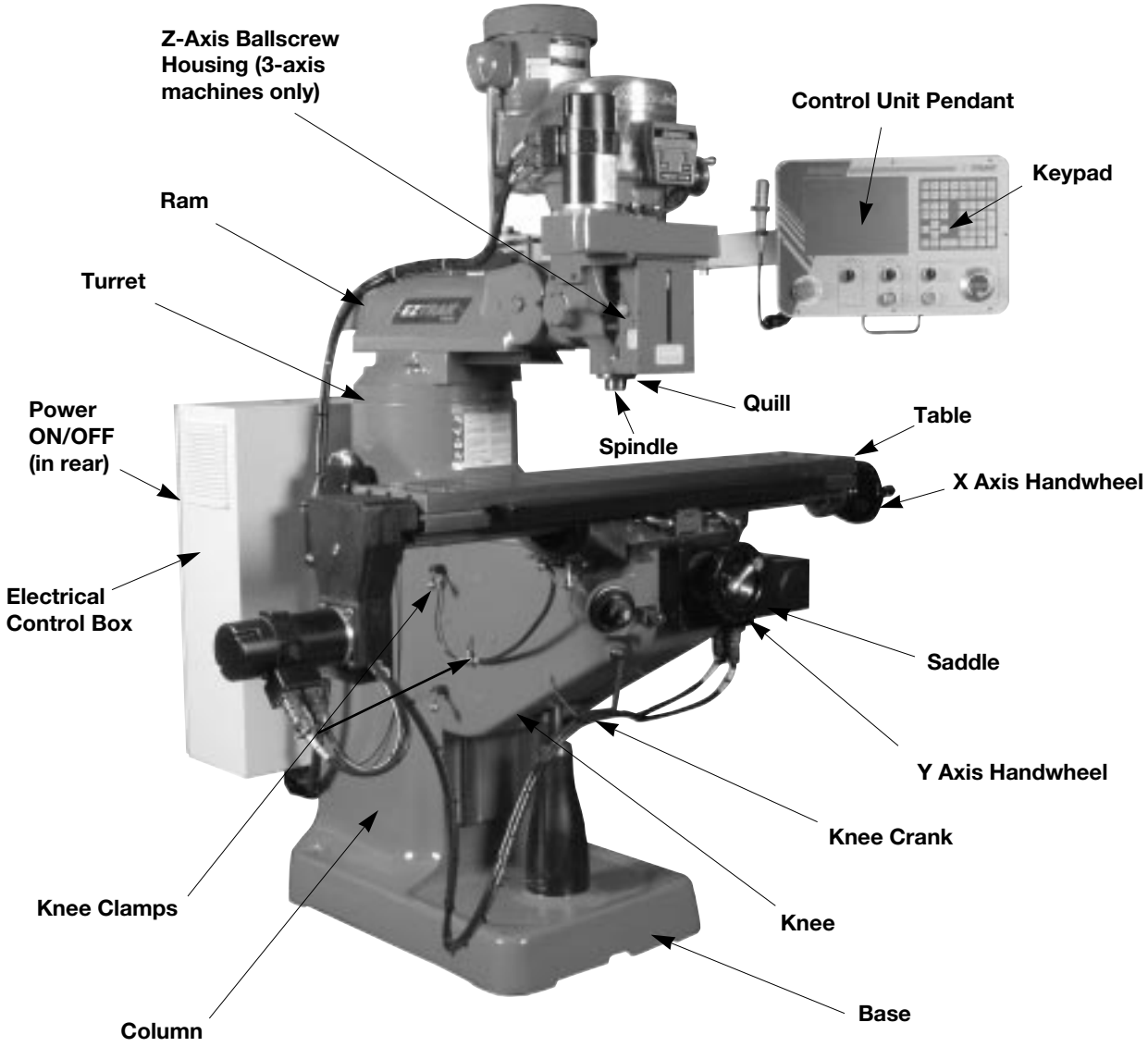


Figure 1-1. EZTRAK 3-Axis Milling Machine major components

1.1 HEAD CONTROLS

Hardware differences between 2-axis and 3-axis EZTRAKs are mainly in the head controls, as illustrated on these pages. Controls which apply only to 2 axis machines are identified with asterisks (Figure 1-2).

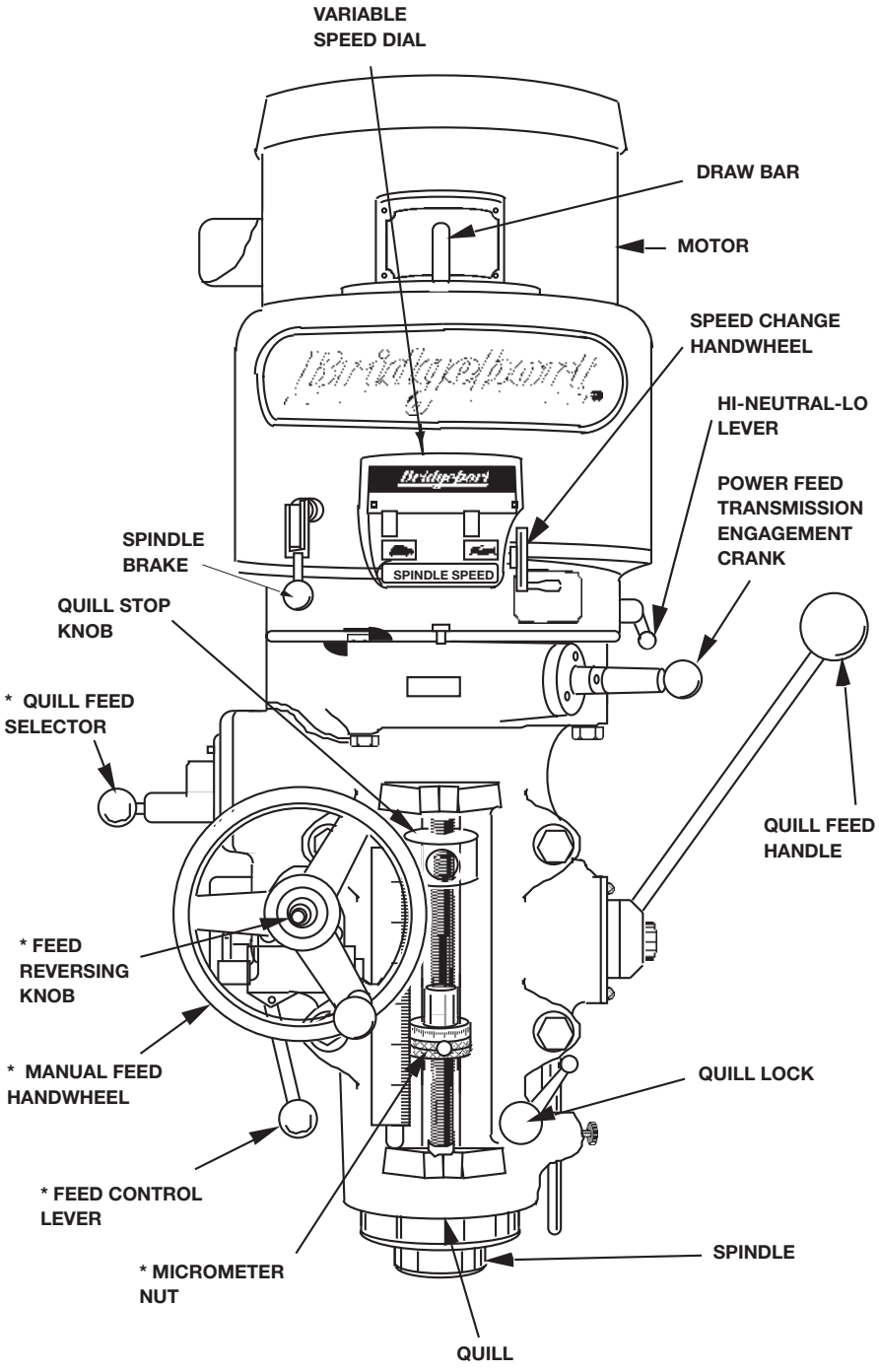


Figure 1-2 Basic EZTRAK 2-Axis Head Controls and Components

In 3-axis configurations, Manual Z- axis feed controls are replaced by an automated feed control within a boxlike quill housing.

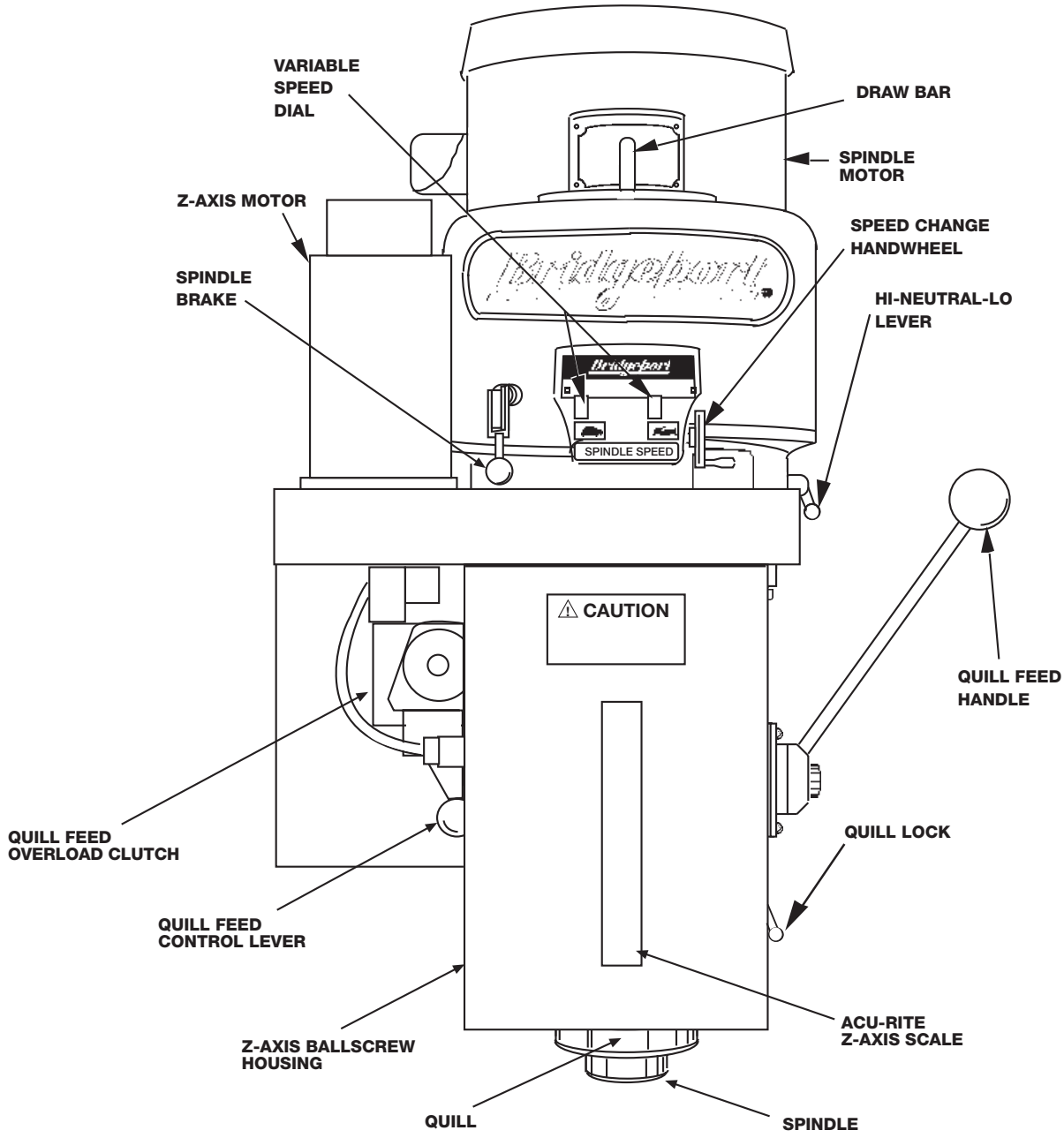


Figure 1-3 Basic EZTRAK 3-Axis Head Controls and Components

1.2 SPINDLE SPEED RANGES

The EZTRAK has two spindle speed ranges: **60-500 rpm (Low range)** and **500-4200 rpm (High range)**. Low rpm is obtained through the Back-Gear drive; high rpm is obtained through Direct Drive.

Three EZTRAK components are used to set spindle speeds: the **Hi-Neutral-Lo Lever** (Figure 1-6) the **Spindle Gear Selector** on the Operator's Control Panel. (Figure 1-7), and the **Speed Change Handwheel** on the Variable Speed Dial (Figure 1-8)

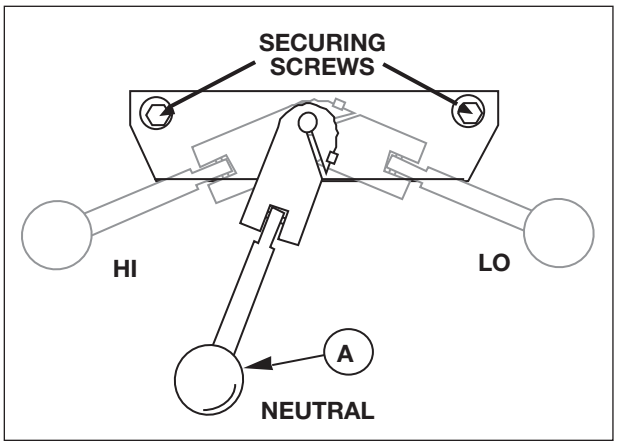


Figure 1-4: Hi-Neutral-Lo Lever

1.2 1 SPEED RANGE SELECTOR (HI-NEUTRAL-LO LEVER) (Figure 1-4)

At the lower right side of the spindle housing is the **Hi-Neutral-Lo Lever** (Figure 1-4). This lever is used to set the speed range for the spindle motor. It has three positions for the spindle motor gearing: LOW, NEUTRAL, and HIGH. Neutral is provided to permit free spindle rotation for indicating and set-up work. Be certain to check the position of this lever when starting up the EZTRAK.

CAUTION

Do not shift the HI-NEUTRAL-LO lever when the spindle is motion!

When shifting to "LO", ***do not force the lever*** if the back gears do not mesh. Hold the lever so the gears are clear of one another, rotate the spindle nose by hand until the gears line up, then put the unit in "LO" (back gear).

In direct drive ("HI" position), the spindle is driven by a tapered tooth clutch. When shifting to "HI", ***do not force the lever*** if the clutch teeth do not mesh. Engage the brake and rotate the spindle nose by hand until the clutch engages.

If the clutch is not meshed tightly, clutch rattle will be heard. This can be corrected by loosening the two securing screws on the lever while in high speed position (Figure 1-7). The clutch spring will automatically adjust the clutch. Tighten the two securing screws.

1.2.3 SPINDLE START BUTTON (Figure 1-5)

This indicator pushbutton starts the spindle, provided the SPINDLE GEAR SELECTOR switch is not OFF. **When this green light is ON, power to the spindle is OFF.** Beware that the spindle may still be moving after power is removed and the green light comes on. You must apply the Spindle Brake to stop the spindle at once.

CAUTION

Be sure the Green Light is ON and the Spindle is STOPPED before attempting to change tools!

Figure 1-5: Spindle Gear Selector and Spindle Start Button on Control Panel

1.2.2 SPINDLE GEAR SELECTOR (Figure 1-5)

This three way switch is used to set the rotation direction of the spindle and to shut the spindle OFF. This switch should be set the same as the SPEED RANGE SELECTOR (HI-NEUTRAL-LO). For example, to make the spindle rotate in the clockwise direction, set the Speed Range Selector to HI and the Spindle Gear Selector to HIGH GEAR.

1.2.4 VARIABLE SPEED DIAL

Figure 1-6 shows the **Variable Speed Dial** which is connected to the **Speed Change Handwheel** . Through the 2 windows the dial indicates the speed range in which machine is operating: 60-500 rpm (low range) or 500 to 4200 rpm (high range). Rotate the Speed Change Handwheel to increase or decrease spindle speed.

NOTE Dial indicator speeds will only be approximate. Belt wear will cause a slight variation in speeds from what is indicated on the dial.

Figure 1-6: Variable Speed Dial with Speed Change Handwheel

1.3 CHANGING SPEED WITHIN RANGE

Start the spindle by pressing the SPINDLE START button, then turn the SPEED CHANGE HANDWHEEL to select the required speed.

WARNING

Do not change the spindle speed unless the spindle motor is running!

1.3.1 CHANGING FROM DIRECT DRIVE (HI) TO BACK GEAR DRIVE (LO)

a) Turn the SPINDLE GEAR SELECTOR switch to OFF to stop spindle rotation.
b) Move the HI-NEUTRAL-LO LEVER to LO (this reverses the spindle rotation).
c) Turn the SPINDLE GEAR SELECTOR switch to LOW GEAR.

1.3.2 CHANGING FROM BACK GEAR DRIVE (LO) TO DIRECT DRIVE (HI)

a) Turn the SPINDLE GEAR SELECTOR switch to OFF to stop spindle rotation.
b) Move the HI-NEUTRAL-LO LEVER to HI.
c) Rotate spindle by hand until the clutches are felt to engage.
d) Turn the SPINDLE GEAR SELECTOR switch to HIGH GEAR.

1.4 SPINDLE BRAKE

To the left of the Variable Speed Dial, on the spindle belt housing, is a short black knob, shown in Figure 1-7. This is the Spindle Brake lever. When the spindle is shut off (i.e., the green light on the Operator's Control Box is ON), this knob can be used to slow the spindle to a stop.

The Spindle Brake lever can be moved in either direction to stop the spindle. Pull the lever toward you or push it away from you and then raise it to lock the brake.

Figure 1-7: Spindle Brake lever.

CAUTION

Be sure that the Spindle Brake is released before starting the motor. The motor can be damaged if the switch is turned on with the brake in the locked position.

1.5 QUILL

The Quill contains the spindle assembly and can be raised or lowered by means of the Quill Feed Handle.

1.6 QUILL FEED HANDLE

This handle is used to raise and lower the Quill manually. The Quill Feed handle may be removed by simply pulling the handle off the Quill Feed Shaft as shown in Figure 1-8. **Remove the handle when using power feed or 3-axis mode.**

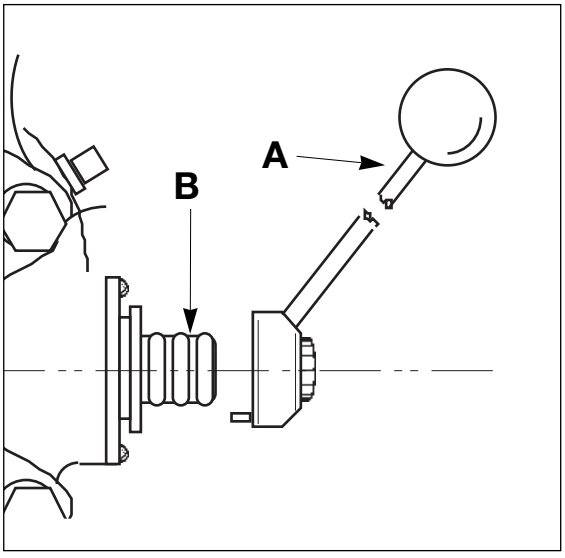


Figure 1-8 Close Up View of the Quill Feed Handle (A) & Shaft (B)

1.7 QUILL FEED CONTROL LEVER

The Feed Control Lever (to the right rear of the Z axis ballscrew housing) engages the overload clutch on the pinion shaft when positioned left, and will stay engaged until either the quill comes in contact with the micrometer adjusting nut, forcing the feed control lever to drop out automatically, or released manually by moving the lever to the right.

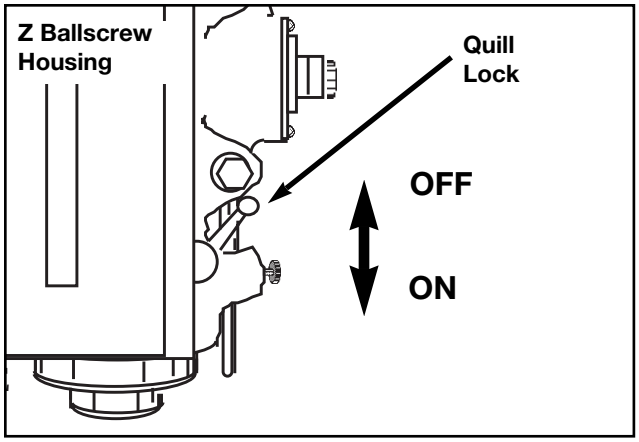


Figure 1-9: Quill Lock

1.8 QUILL FEED OVERLOAD CLUTCH

The feed control overload clutch is factory set to hold up to 200 lbs. down pressure on the Quill. The *Feed Control* Lever must be engaged in order to use manual feed controls.

Do not tamper with the feed control

CAUTION

overload clutch.

1.9 QUILL LOCK

The Quill Lock is a friction lock used when quill is in stationary position such as milling operations. It is recommended that this lock be used whenever quill movement is not desired.

1.10 DRAWBAR

Figure 1-10 illustrates an enlarged view of the top of the Drawbar. When tightening or loosening the Drawbar, it is necessary to lock the spindle. To accomplish this, apply the Spindle Brake until it binds, then raise the Quill Feed Handle to lock it in place.

The drawbar has a 7/16-20 right hand thread and should be tightened with a normal amount of pressure, using the wrench furnished with the machine. To loosen the collet, back off the drawbar ; if the collet does not open immediately, give the knob on top of the drawbar a slight tap. The spindle has a non-sticking taper and the collet should release readily.

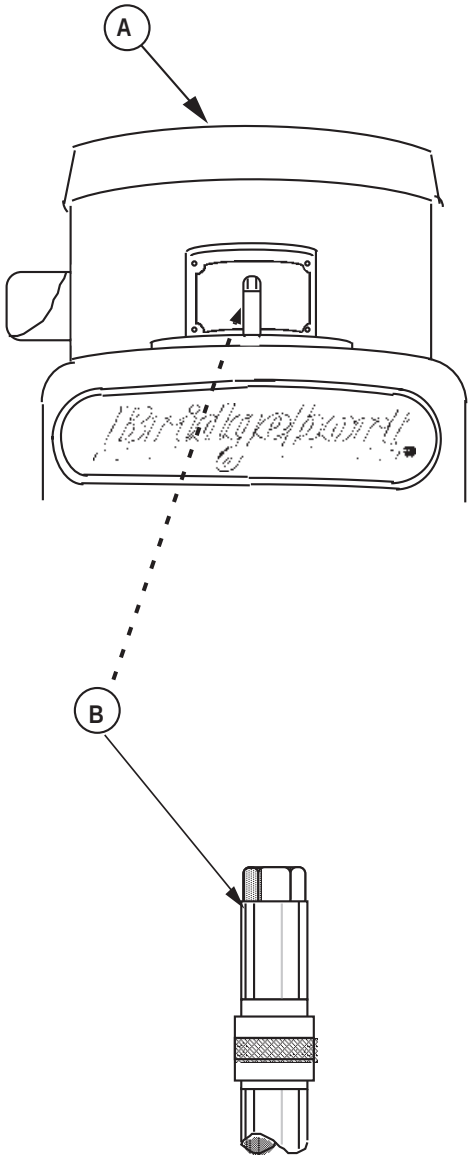


Figure 1-10: Motor (A) and Drawbar (B).

1.11 SWIVEL HEAD

The top of the EZTRAK head can be swiveled by loosening the 3 locking nuts (A) shown in Figure 1-12 and using the following procedure.

WARNING

To prevent personal injury or damage to machine, do not remove the three locking nuts after loosening.

1) Loosen the 3 locking nuts BUT DO NOT REMOVE.

2) Swivel the head to the required angular setting.

CAUTION

Incorrect spline alignment can be caused by unequal tightening of the locking nuts, causing fluctuation of the quill feed which can be felt through the sensitive feed handle. Call your local Sales & Service Center before attempting this procedure.

3) Tighten the 3 locking nuts snugly before final tightening. Run the spindle to give correct spine alignment, then retighten locking nuts securely.

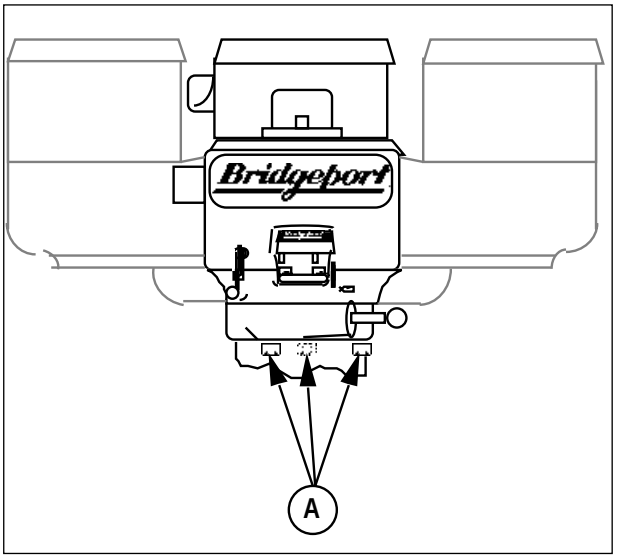


Figure 1-11: Three Locking Nuts (A) to loosen in order to swivel the EZTRAK, head.

1.12 TURRET

The EZTRAK® turret can be swiveled using the following procedure.

WARNING

DO NOT REMOVE BOLTS FROM TURRET.

a) Use Bridgeport wrench to loosen the 4 bolts shown in Figure 1-13.

b) Index to the required setting.

c) Lock the 4 bolts to 47 ft. lbs.

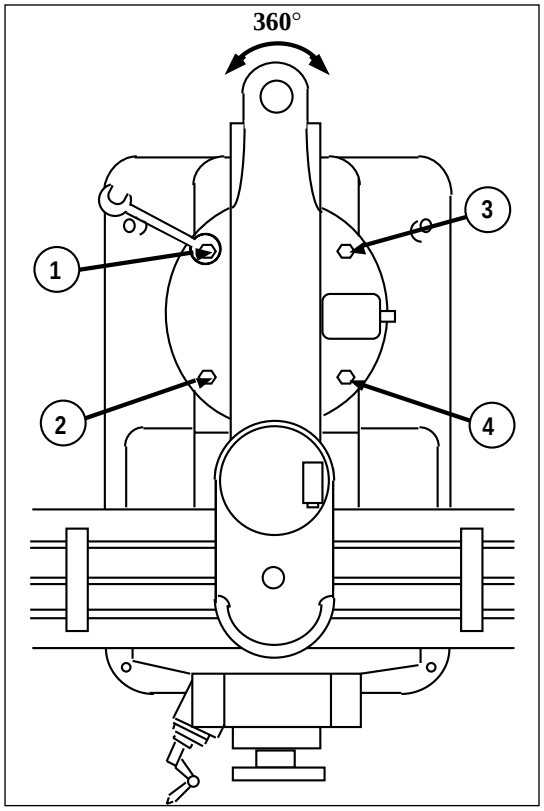


Figure 1-12: Top View of EZTRAK, showing the 4 bolts to be loosened to swivel the turret.

1.13 RAM ADJUSTMENT

As shown in Figure 1-13, the Ram portion of the EZTRAK® can be slid front to back by loosening several bolts as follows:

a) Use Bridgeport wrench and loosen bolts 1 and 2.

b) Use wrench to move the slide to the desired position using bolt 3.

c) Retighten bolts 1 and 2 starting with the rear bolt.

NOTE: It is recommended that on heavy milling work, the head should be kept as close to the column as possible, where maximum rigidity is obtained.

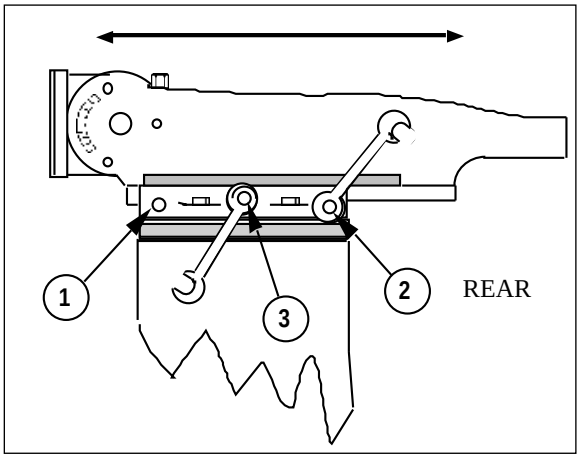
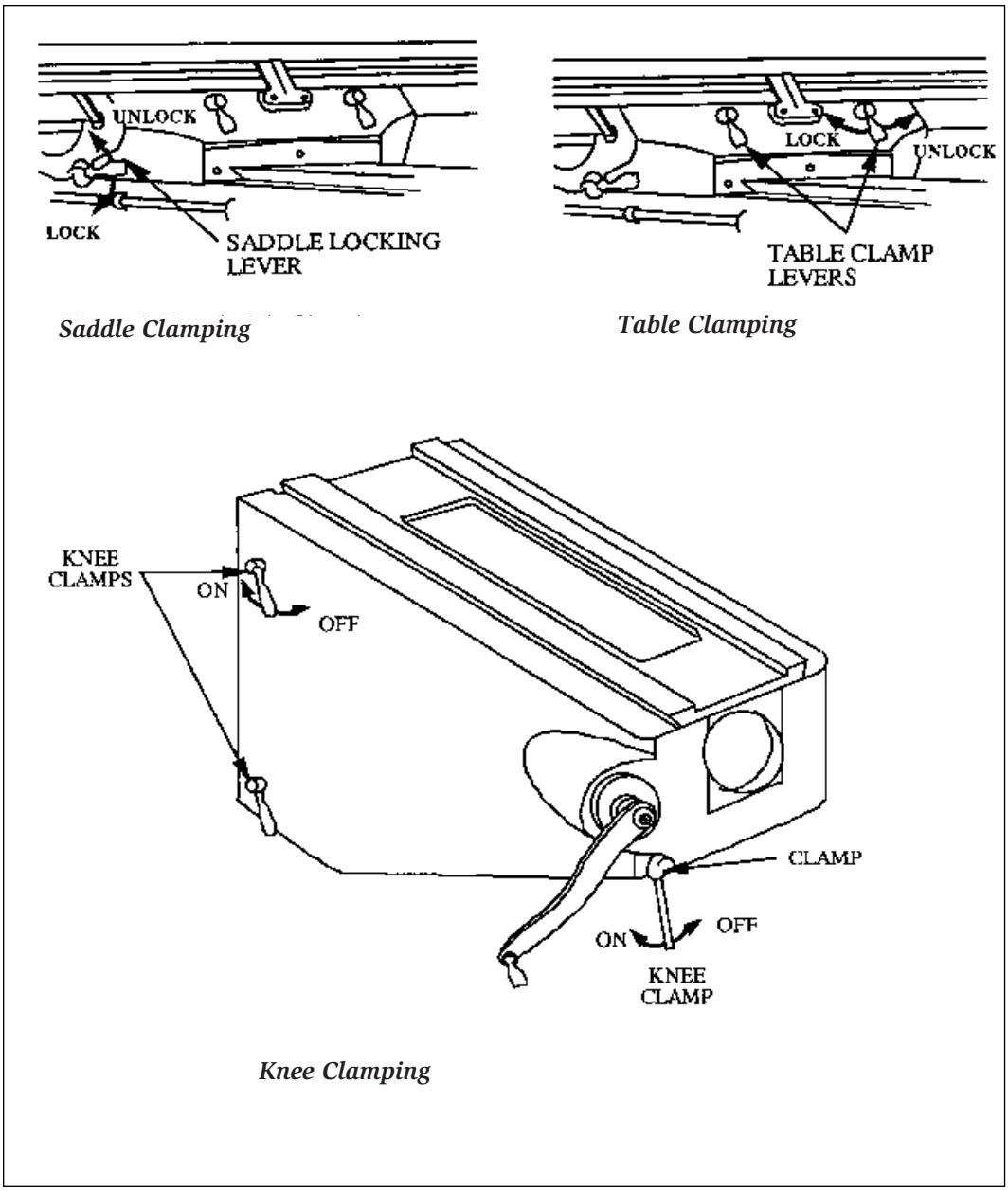


Figure 1-13: Ram Slide Adjustment



Figures 1-14: Saddle Clamping, Table Clamping, and Knee Clamping Locations for EZTRAK I Machine

1.14 SADDLE CLAMPING

Figures 1-14 and 1-15 illustrate the locations of saddle clamps for the EZTRAK® & EZTRAK® II machines. When milling with longitudinal table feed only, it is advisable to clamp the knee to the column and the saddle to the knee to add rigidity to these members and provide for heavier cuts with minimum vibration. The saddle locking lever is located on the left-hand side of the saddle for the EZTRAK® II. Excessive pressure can cause slight table bind. Use moderate clamping pressure, as this will hold the saddle sufficiently.

NOTE: Saddle Clamp should not be used when Power to axis drives is ON.

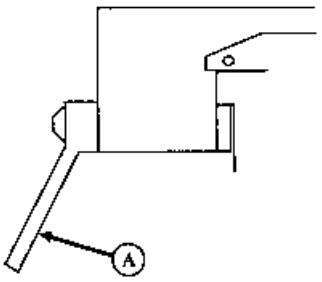
1.15 TABLE CLAMPING

Figures 1-14 and 1-15 illustrate the locations of table clamps for the EZTRAK® & EZTRAK® II machines. Table clamp levers are located on front of saddle for the EZTRAK® II, and should always be clamped when longitudinal movement is not required.

NOTE: Table Clamp should not be used when Power to axis drives is ON.

1.16 KNEE CLAMPING

Figures 1-14 and 1-15 illustrate the locations of knee clamps for the EZTRAK® & EZTRAK® II machines. Knee clamping levers are at the left side of the knee and front of knee for the EZTRAK®, and in the front of the column base for the EZTRAK® II. Leave clamped at all times unless raising or lowering the knee using the Knee Crank Handle.



Saddle Clamping

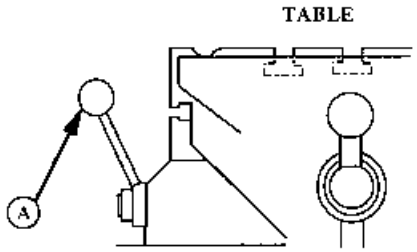
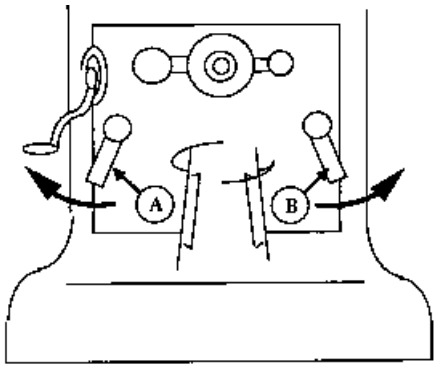


Table Clamping



Knee Clamping

Figure 1-15: Clamp locations for EZTRAK II Machine.

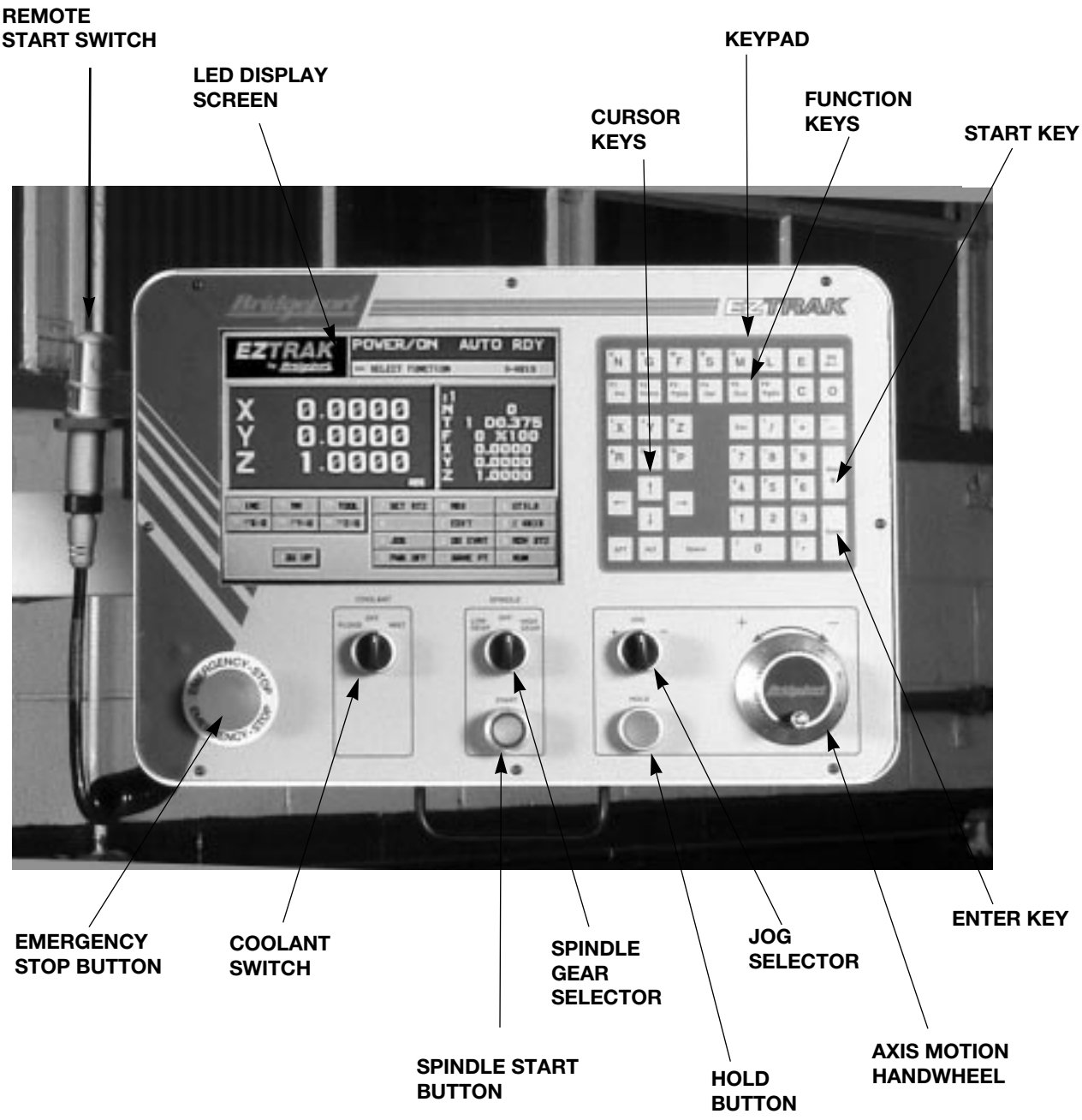


Figure 1-16: Control Panel

1.18 FLAT PANEL CONTROLS:

LCD DISPLAY SCREEN
The screen on the Control Panel displays all of the information necessary to run the EZTRAK, such as tool number and diameter, feedrate, and program information.

EMERGENCY STOP
This red mushroom switch is used to shut down all power to the spindle and axis drives in the event of an emergency. When the E-Stop button is pressed, the computer screen displays an ALARM condition in the top left corner of the screen. To clear the alarm condition, pull out the EMERGENCY STOP button by turning it, then follow the instructions on the screen to continue. DO NOT RESTART THE EZTRAK BEFORE PULLING OUT THE EMERGENCY STOP BUTTON. ***CAUTION:*** ***Pressing the Emergency Stop button or Spindle Off selector removes power from the spindle motor but does NOT apply the spindle brake. The brake must be activated manually.***

COOLANT
This three way switch (FLOOD-OFF-MIST) activates flood or spray mist coolant functions.

SPINDLE GEAR SELECTOR
This three way switch (LOW GEAR-OFF-HIGH GEAR) is used to set the rotation direction of the spindle and to shut the spindle OFF. It should be set the same as the SPEED RANGE SELECTOR located on the lower right side of the spindle housing.

SPINDLE START
This indicator pushbutton starts the spindle, provided the Spindle Gear Selector is not turned to OFF. When this green light is ON, power to the spindle is OFF. ***CAUTION:*** ***Never put hands near the spindle unless the spindle light is ON.***

JOG SELECTOR (+/-)
AXIS MOTION HANDWHEEL
In JOG mode, these two controls are used to position the table by moving the X and Y axes separately. After selecting the axis you wish to move, turn the JOG SWITCH to plus (+) or minus (-) to jog the axis in the positive or negative direction.

HOLD
This pushbutton interrupts automatic operation. The START button resumes operation.
CAUTION: ***The HOLD switch will not shut off the spindle or coolant. It is not the Emergency Stop.***

REMOTE START SWITCH
This is a hand held switch attached to a flexible cord at the left side of the operator's control panel. It can be used in place of the START button on the EZTRAK keyboard when the operator is prompted to press the START button.

KEYPAD

The keypad is used to enter commands to the EZTRAK. Each key is assigned a command or function, which is shown on the screen. The function of each key may change as the mode of operation changes. See the chapter on Basic Operation of the EZTRAK.

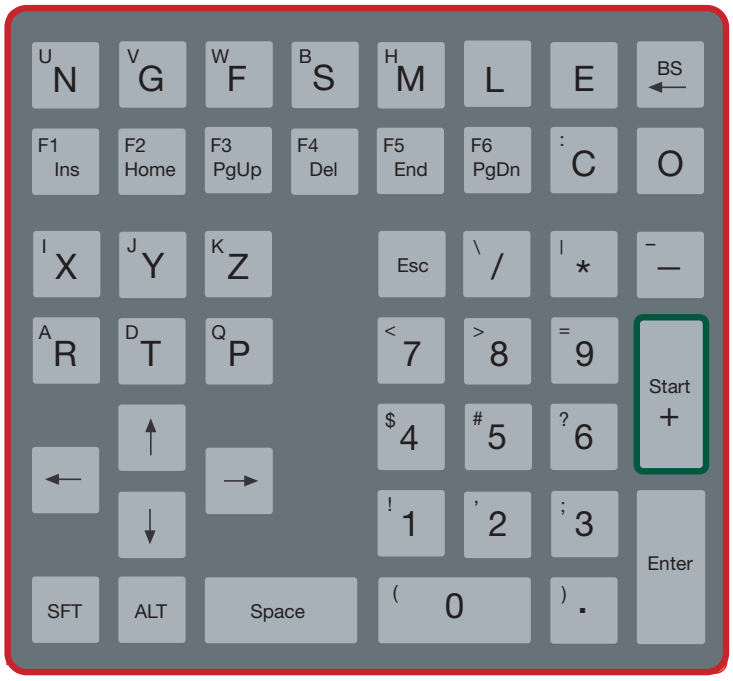


Figure 1-17: Keypad on the Operator Control Panel

Key Descriptions

ALT The Alternate key accesses the menus for G-Code Editing.

SFT Shift enables use of the letters, symbols or functions printed in the upper left corners of the keys.

BS Backspace key.

START+ Runs a part program.

ENTER Used for entering commands.

SPACE Inserts space between text elements.

INS Insert text.

HOME Places the cursor at the top of the page.

PG UP Page Up

PG DN Page Down

END Places cursor at the end of a line.

DEL Delete

ESC Escape exits the part program.

Arrow Keys Moves the cursor through a program.

1.19 EZTRAK COMPUTER HARDWARE

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

The EZTRAK has a standard throughput of 250 data blocks per second, with 256 kilo-bytes of high speed memory on the BMDC board for part program storage. An 8 Meg. Disk On Module (C:\) is used for part program and operating system storage. An optional larger Disk On Module (32 Meg) or Hard Drive is available for additional data storage.

1.20 EZTRAK COMPUTER SOFTWARE

The software which drives the EZTRAK is best described in terms on the hardware that runs it. All of this software is loaded from the Disk On Module at startup.

- **PC AT.** The software used on the PC AT consists primarily of the Microsoft DOS v. 6.20 operating system, and EZTRAK.EXE, the operator interface software.

1. DOS (Disk Operating System). DOS is a collection of routines that perform basic computer tasks such as starting the computer (booting up), moving data to and from disks and peripheral devices, and managing and allocating memory space.

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

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

- **BMDC** Motor Drive Controller.

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

2. SYSTEM DISK. A Disk On Module (C:\) stores all of the system software for the EZTRAK, as well as any part programs that the user may have created and stored. This disk is essential to running the EZTRAK.

3. BACKUP SYSTEM DISK. A floppy diskette provided with the machine contains a backup copy of all the system software and machine parameters (like backlash and lead comp) for the EZTRAK. In the unlikely event of a disk crash, the backup disk can be used to restore the system.

NOTE: It is highly recommended that you keep two copies of the SYSTEM DISK in a safe place. It is also recommended that you back up your part programs to a floppy disk once every 6 months.

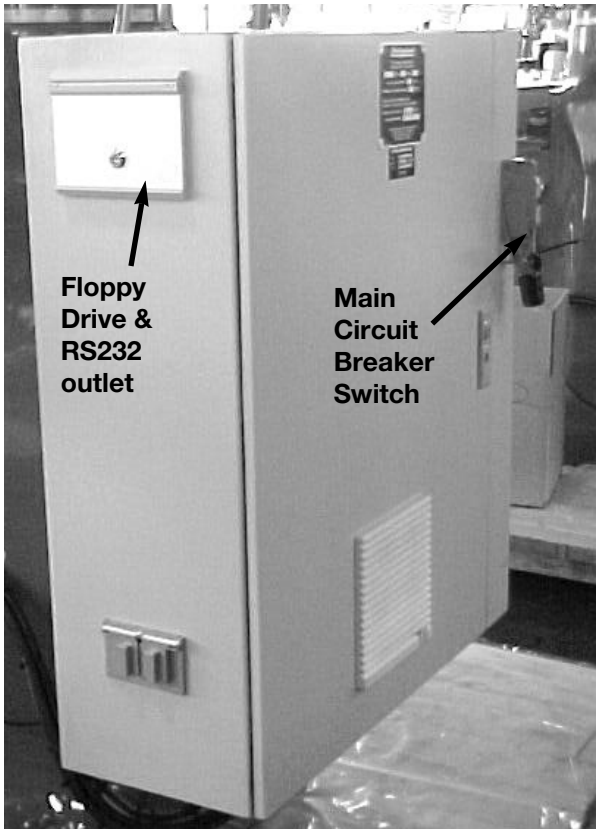


Figure 1-18. Electrical Cabinet

Figure 1-19. Floppy Drive and RS232 outlet (with cover) on left side of electrical cabinet.

Chapter 2
STARTING UP THE EZTRAK

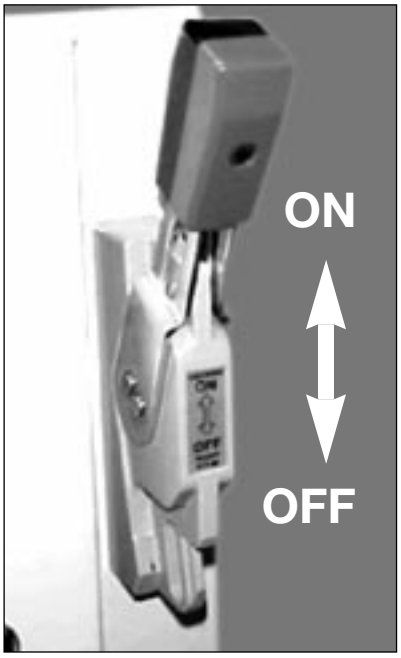
The following 2 steps *must* be taken before the EZTRAK® can be used.

1) **Turn the Power ON;** and
2) **Home the Axes.**

The 3-axis EZTRAK can only be homed in 3-axis mode. Therefore, whether you intend to use the machine in manual (2-axis) or automatic (3-axis), you must first engage the 3-axis mode after startup, as decribed below.

2.1 TURNING ON THE EZTRAK

On the back of the EZTRAK's electrical cabinet is a large switch. This is the Power Switch (MAIN DISCONNECT) for the machine. It turns on power to the computer as well as to the drive motors and spindle motor. Turn the Power Switch to the ON position as shown by the arrow. You will feel a solid click when the power is turned on.



NOTE : It is not a good idea for anyone to stand behind the machine while it is in operation.

Figure 2-1 MAIN DISCONNECT at the back of the machine.

As the computer starts up, you will see various messages flash across the LCD display on the operator's panel. These messages are used for loading software and may be disregarded during normal startup. When full power is reached, the screen shown in Figure 2-2 is displayed.

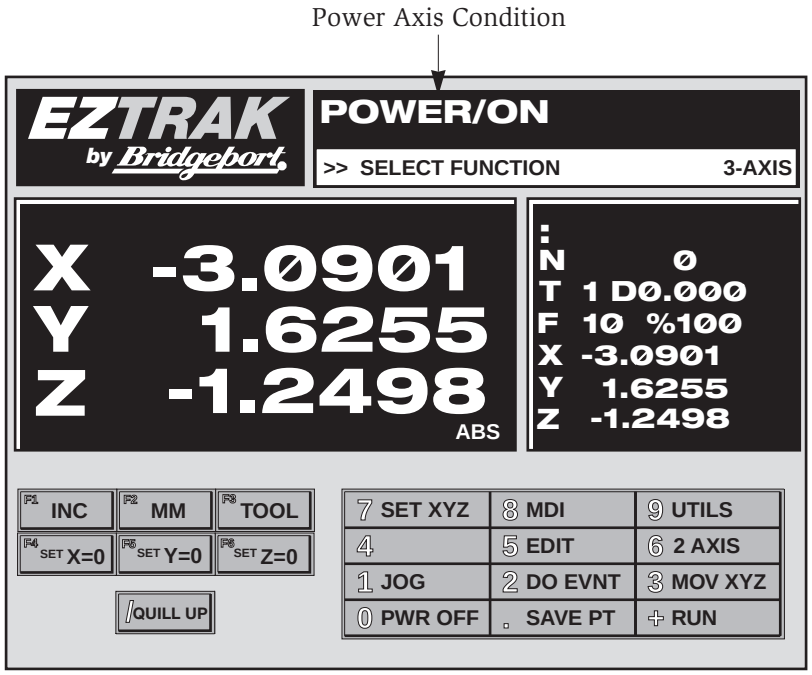


Figure 2-2. BASIC OPERATIONS SCREEN. Note: [6 3 AXIS] and [/QU UP] do not apply to 2-axis machines

There are 2 power modes for the EZTRAK®, Power ON & Power OFF. **The Power OFF Mode is the Manual Mode,** discussed further in Chapter 3 (Basic Operation). Most EZTRAK functions will be performed in the **Power ON Mode.**

NOTE :The upper right corner of the screen usually displays the Power Mode of the machine.(Figure 2-2).

POWER/OFF does not mean that power to the spindle or control computer is off!

Notice at the top left of the screen the message which reads 'EZTRAK NOT HOMED - HIT [MOV ABS]'. This means that the EZTRAK® must go through the process of homing the axes (see the next section) before the machine is ready to operate.

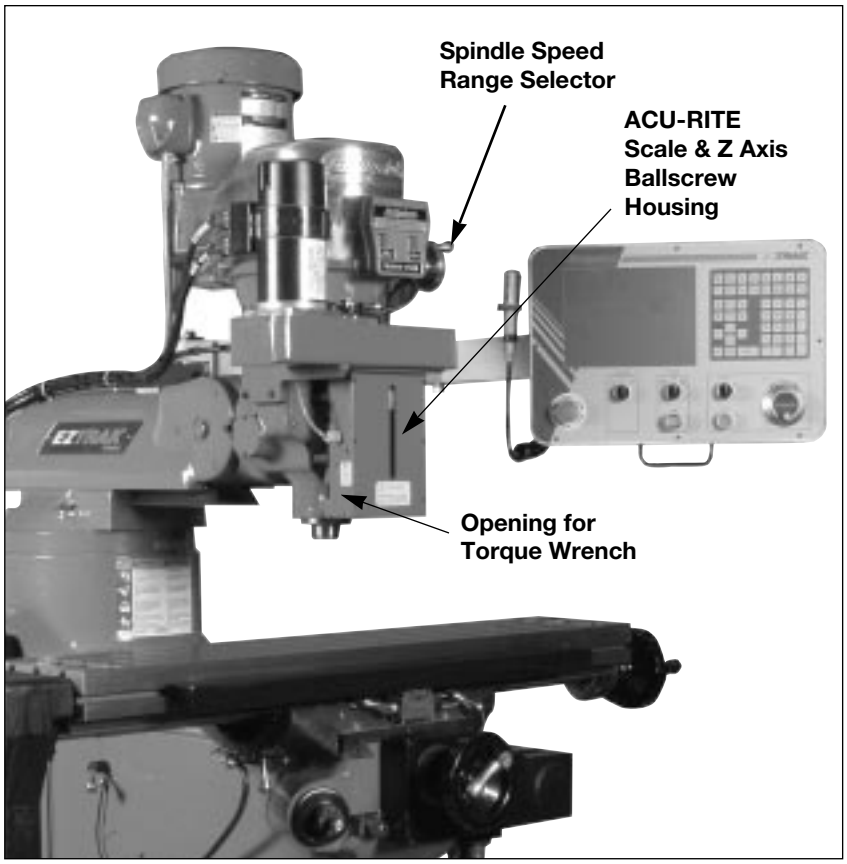


Figure 2-3. A basic EZTRAK 3-axis machine indicating the Acu-Rite scale housing, and the ballscrew clamp opening.

2.2 ENGAGING 3 AXIS MODE

As the EZTRAK starts up, a message will appear at the top of the screen:

> **Position Quill WITH THE HANDLE to furthest Z minus travel limit then tighten clamp on ballscrew by turning in CW direction to engage 3-axis mode. Press any key when done.**

This message will always appear at startup, whether the machine is already in 3-axis mode or not. It is a prompt to the operator to ensure that the machine is in 3-axis mode before continuing to the next step of homing the axes.

Changing the operating mode of the machine from 2-axis to 3-axis, or vice-versa, requires an adjustment of a clamp on the Z-axis ballscrew. The clamp and the ballscrew are inside the Ballscrew Housing mounted on the front of the machine's head. Access to the clamp is through a small hole on the left side of the ballscrew housing, covered by a hinged metal flap (Figure 2-3). The clamp must be torqued to 22 ft lbs using a 1/4" Allen wrench attached to a Torque Wrench handle, which are provided with the machine. You will be prompted to make this adjustment by messages on the control panel screen.

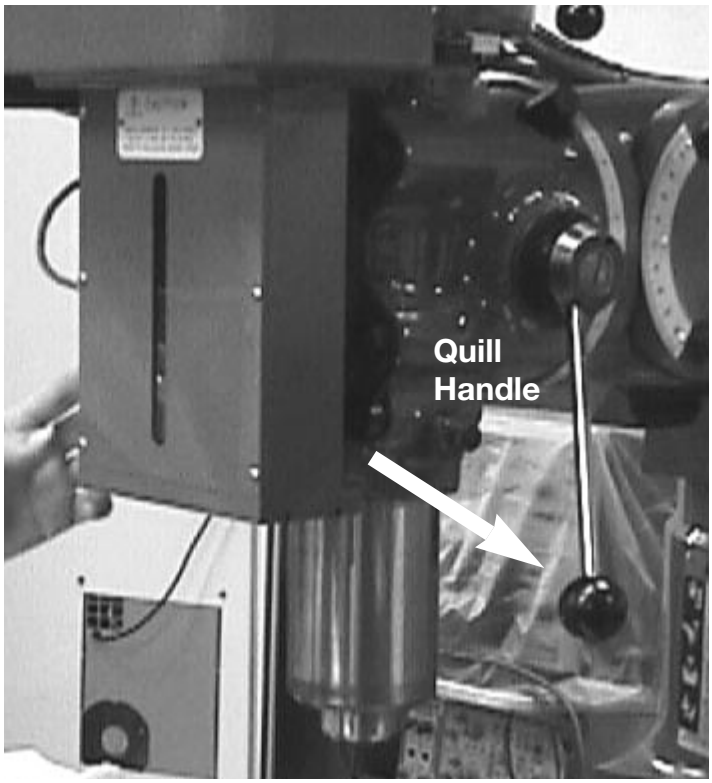


Figure 2-4a:
Manually lower quill to its limit of travel, using the detachable Quill Handle.

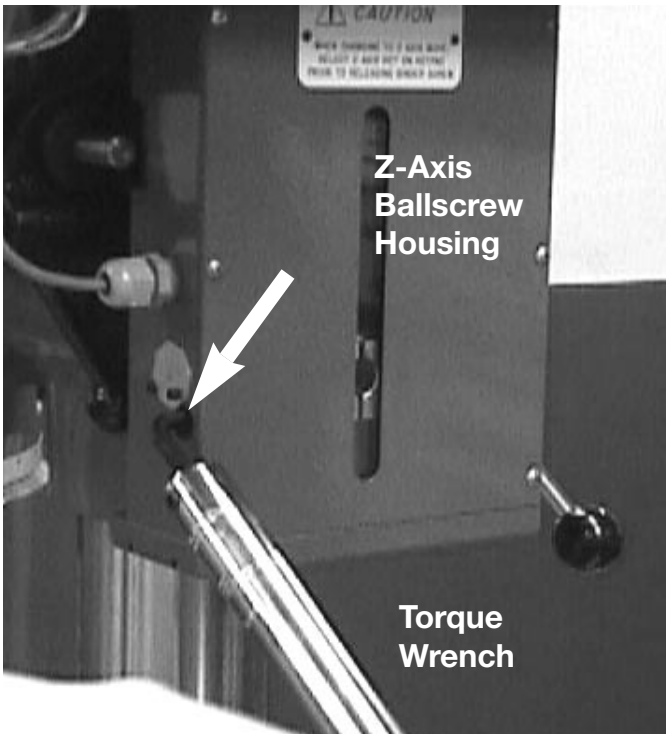


Figure 2-4b:
Insert 22 ft lb Torque Wrench into outlet on the left side of Ballscrew Housing to adjust ballscrew clamp.

To change from 2-axis to 3-axis mode:

1. Attach the removeable Quill Handle and manually crank the Z-axis all the way down to its minus limit.

1. Open the small flap marked 1/4" on the left side of the housing.

2. Insert the 1/4" Torque Wrench to engage the ballscrew clamp.

3. Torque the ballscrew clamp in a downward or Clockwise (CW) direction to 22 ft lbs. The wrench will click when the proper torque is obtained.

4. Press ENTER on the Control Panel.

At this point a Warning message will flash on the screen:

WARNING, IS THE LOCKING MECHANISM LOCKED TO THE PROPER TORQUE USING THE TORQUE WRENCH PROVIDED? DO NOT PROCEED IF THIS HAS NOT BEEN DONE.
REMOVE QUILL HANDLE BEFORE STARTING.
PRESS [+] TO CONTINUE.

Mechanical problems may result if the torquing procedure has not been done before going onto the next step! When you are certain that the locking nut adjustment has been made properly, remove the Quill Handle and hit the + key on the keypad.

The machine is now operable in 3-axis mode, and you may proceed to the next step of homing the axes.

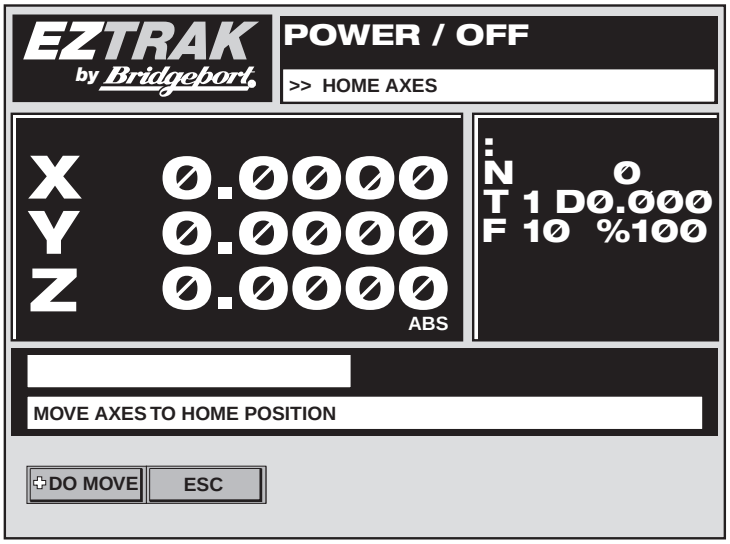


Figure 2-5 . After the machine is first powered up, you will need to apply power to the axis by pressing the 3 key [MOVE ABS], then the + key [DO MOVE] to move the axis. You will then be given the opportunity to assign X, Y, & Z values in the [MOV ABS] screen as shown in Figure 2-6.

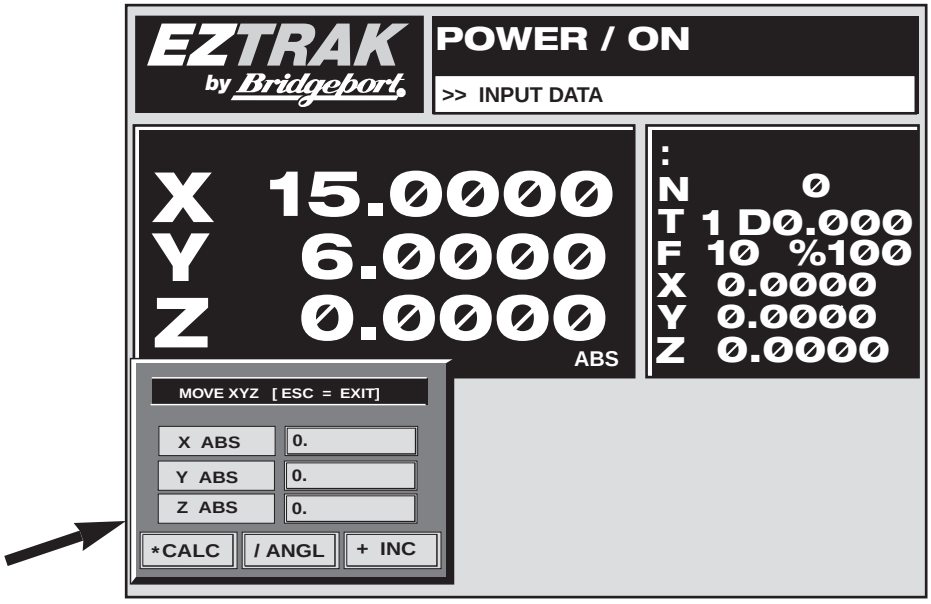


Figure 2-6. [MOV ABS] screen.

2.3 HOMING THE AXES

The EZTRAK needs to find the limits of the worktable travel each time the machine is turned on. This process is called homing the axes, and it can be done only in 3 axis mode.

After you have adjusted the clamp on the Z-axis ballscrew, and pressed the + key on the control panel, this message will appear:

EZTRAK NOT HOMED, HIT MOV XYZ.

At this point, the **3** key on the keypad has the function **MOV XYZ.** Press this key to apply power to the axis drive motors.

WARNING

For safety, fold the handles on the handwheels on the front and right sides of the machine before powering the axis drives. It can be dangerous to leave these unfolded while the machine is running in automatic mode. Also, be sure to remove the Quill Handle before running the machine in automatic mode.

Figure 2-7: Folding handle on axis handwheel.

After you press the **3 (MOV XYZ)** key, the screen display prompts you to press the + key to home the axes. When you hit the + key, the Z axis goes up until it finds the trip switch inside the Z axis ballscrew housing. Then the table begins to move: the Y-axis moves forward and the X-axis moves left. When they find the trip switches under the table, the X and Y axes stop moving, and the limits of the table are set. This also resets the X, Y and Z coordinates, so that the last saved origin is active.

The screen now displays a box labeled **MOVE XYZ ABS,** listing the coordinates for each axis. The display will show a blank in each of the coordinate boxes. You may enter new coordinates or escape. If a coordinate box is left blank, there will be no movement for that axis. (Note that pressing ENTER three times has the same effect as pressing the ESC key.)

If numbers are entered, the table and/or quill will move. When the move is completed, the display returns to the BASIC OPERATIONS screen.

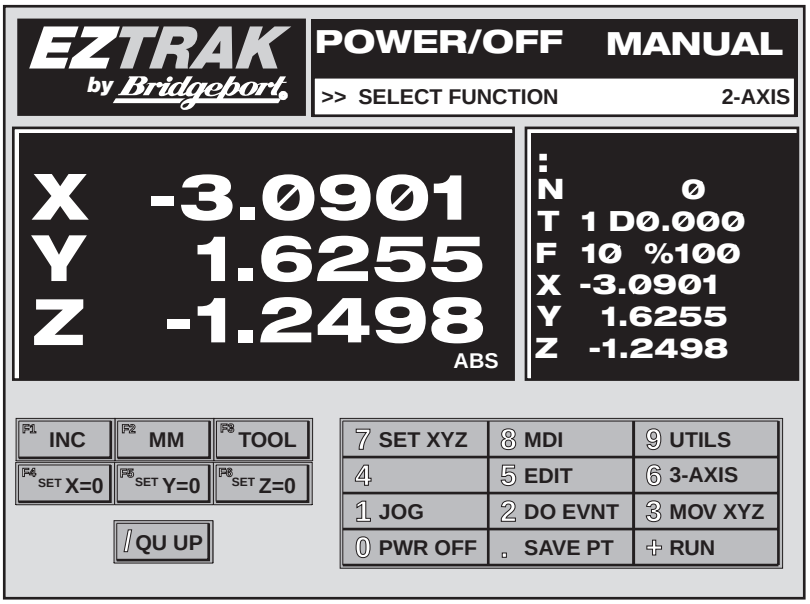


Figure 2-8 BASIC OPERATIONS screen after pressing the 0 key [0 PWR OFF]. You can now move the X & Y axis handwheels to adjust the axis positions manually.

2.6 CHANGING FROM 3 AXIS TO 2 AXIS MODE (MANUAL Z MOVEMENT)

Once the machine is homed in 3-axis mode, a message will appear on the screen prompting you to proceed if you wish to go back to 2-axis mode:

JOG THE Z-AXIS TO ITS FURTHEST MINUS TRAVEL LIMIT TO ENGAGE THE 2 AXIS MODE.

PRESS ANY KEY WHEN DONE.

To change from 3-axis to 2-axis mode:

1. JOG the Z axis to its furthest minus limit, as follows. (The machine will not let you continue until you have done so.)

   a. Press 1 JOG.

   b. Press F3 JOG Z.

   c. Turn the JOG KNOB all the way to minus, making sure the Quill has room to move all the way down (4 1/2") without obstruction.

2.Press 0 EXIT.

3. Press 6 [2 AXIS].

This message will appear on the screen:

LOOSEN CLAMP ON BALLSCREW BY TURNING IN CCW DIRECTION TO RETURN TO 2 AXIS MODE.

4. Insert the torque wrench through the hole in the left side of the ballscrew housing, and turn it upwards or Counterclockwise (CCW) 90 degrees.

5. Press ENTER.

2.7 POWER OFF - MANUAL MODE

Press the **0** key **[0 PWR OFF]** to turn off power to the axis drives. Look at the upper corner of the screen as shown in Figure 2-7. The message should now read **'POWER / OFF MANUAL'.** This is the *Manual Mode*. This means that the table can now be moved by turning the handwheels at the front and right side of the worktable.

NOTE: POWER/OFF does not mean that power to the spindle or control computer is off!

Try moving the worktable in each direction, using the X and Y axis handwheels. Note that on the screen the large display numerals showing the X Y and Z coordinates of the table change as you move the handwheels. The X coordinate changes as you move the handwheel at the right side of the table, and the Y coordinate changes as you move the handwheel at the front of the worktable.

NOTE : When you reverse direction using the handwheels, the axes coordinates are not immediately updated. This occurs because the system must compensate for the backlash which is inherent in the ballscrew rotation.

2.8 TURNING OFF THE EZTRAK

Before you shut down the EZTRAK, be sure there are no disks in the A:\ drive (which is located in the rear of the machine), then turn the Power Switch to the OFF position.

The only time you will need to have a disk in the EZTRAK *before* you start up is when you are updating the system software. Follow instructions included with future system updates for proper installation procedures.

NOTE : Before turning the EZTRAK off - whenever possible - DO NOT leave the table in the extreme left or right position. It is good practice to leave the table approximately at the center position.

Chapter 3
BASIC OPERATION

3.1 CONTROL PANEL DISPLAY

When power to the EZTRAK is turned on, the BASIC OPERATIONS screen appears on the flat panel display. From this screen you can access all operational modes of the EZTRAK. Besides information on the operational status of the machine, such as axes positions, current tool number and diameter, etc., the screen displays available functions which are referenced to keys on the Operator's Control Panel.

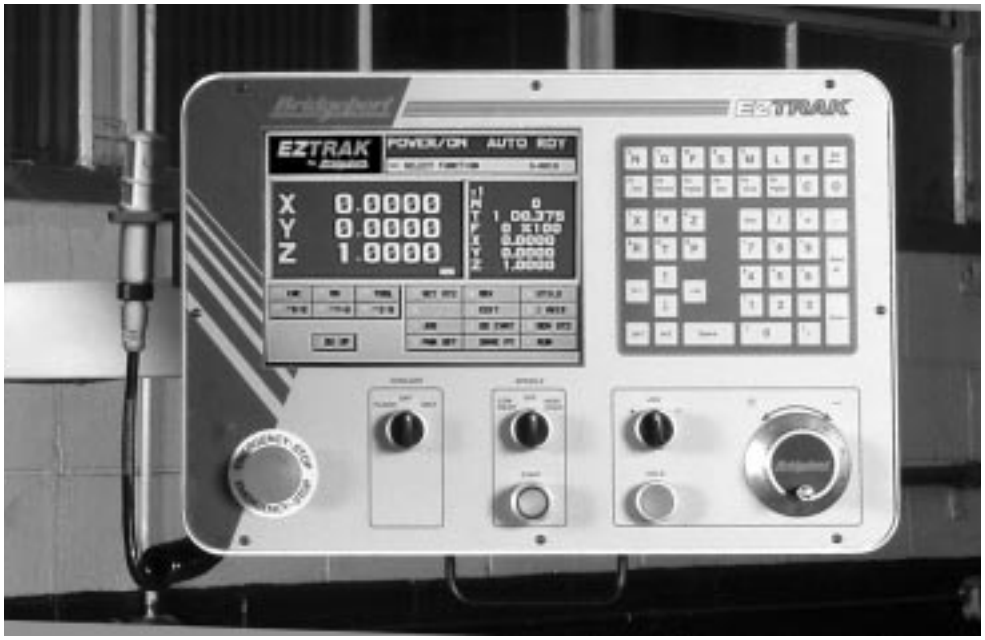


Figure 3-1 Basic Operation Screen

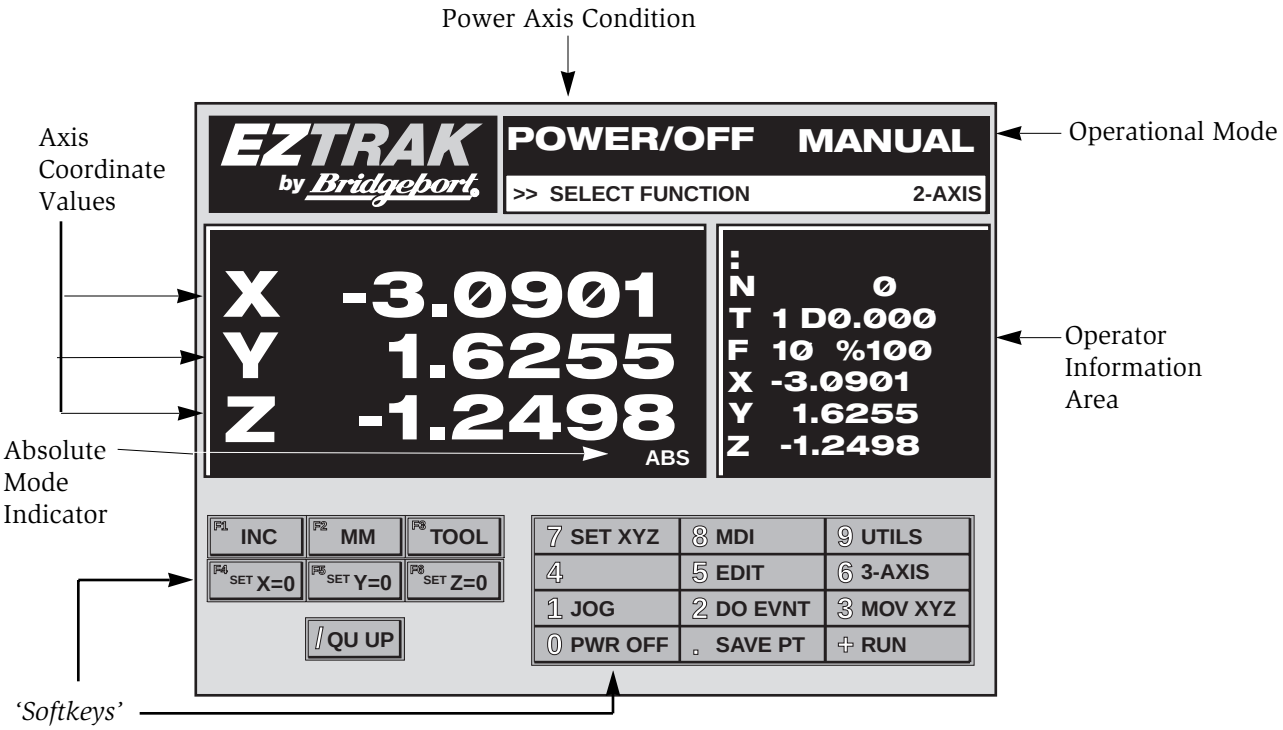


Figure 3-2: BASIC OPERATIONS SCREEN. This figure illustrates the major components of the flat panel screen display.

3.2 READING THE CONTROL PANEL DISPLAY

Figure 3-2 is the BASIC OPERATIONS screen. Down in the lower right corner of the large XYZ display is the indicator for either **Absolute** or **Incremental** Mode.(ABS or INC). The screen shown in Figure 3-2 is in **Absolute** Mode, with Incremental Mode (INC) available as an option at function key F1. In Incremental mode, Absolute (ABS) would be available at F1. **The F1 key** allows you to shift back and forth between **Absolute (ABS)** and **Incremental (INC)** modes.

When the power switch at the back of the machine is turned on, the EZTRAK® defaults to the **Absolute** Mode. (Absolute and Incremental modes are described in Chapter 4 (Manual DRO and Jog) and in Appendix A (Axes and Coordinates.) To the right of the large XYZ display is a smaller box with a similar set of XYZ coordinates at the bottom. This area gives the operator several important pieces of information.

: This identifies the program (i.e., TEACH.PGM) or program number (i.e., 123.PGM) that is currently being run. If no program is loaded into memory currently this area is blank.

N This shows what line number the EZTRAK® is currently executing. If no program is being executed this area also appears blank.

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

D The programmed diameter of the current tool is shown here.

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

% The override percentage of the programmed feedrate is shown here. The override can be increased up to 150%, or decreased to 0%, of the programmed feedrate. For example, if the programmed feedrate is

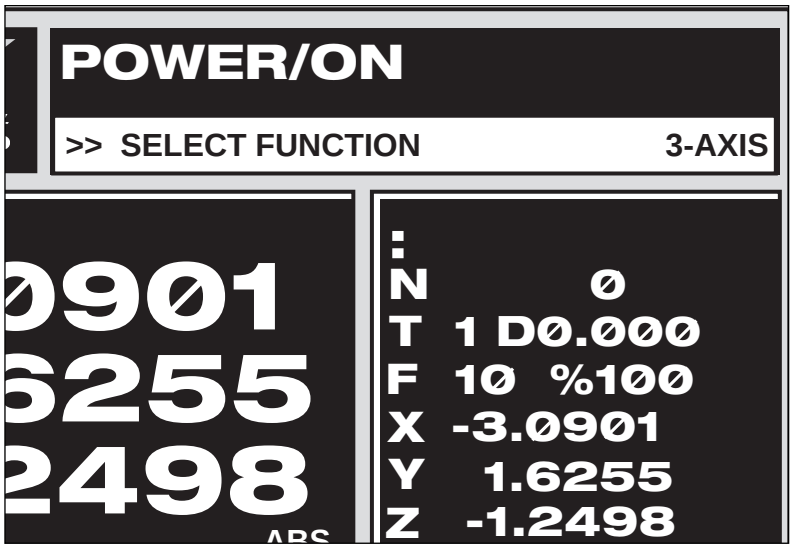


Figure 3-3: BASIC OPERATIONS SCREEN. indicating Power Condition.

10 inches per minute and the % is at 120, the feedrate is increased to 12 inches per minute.

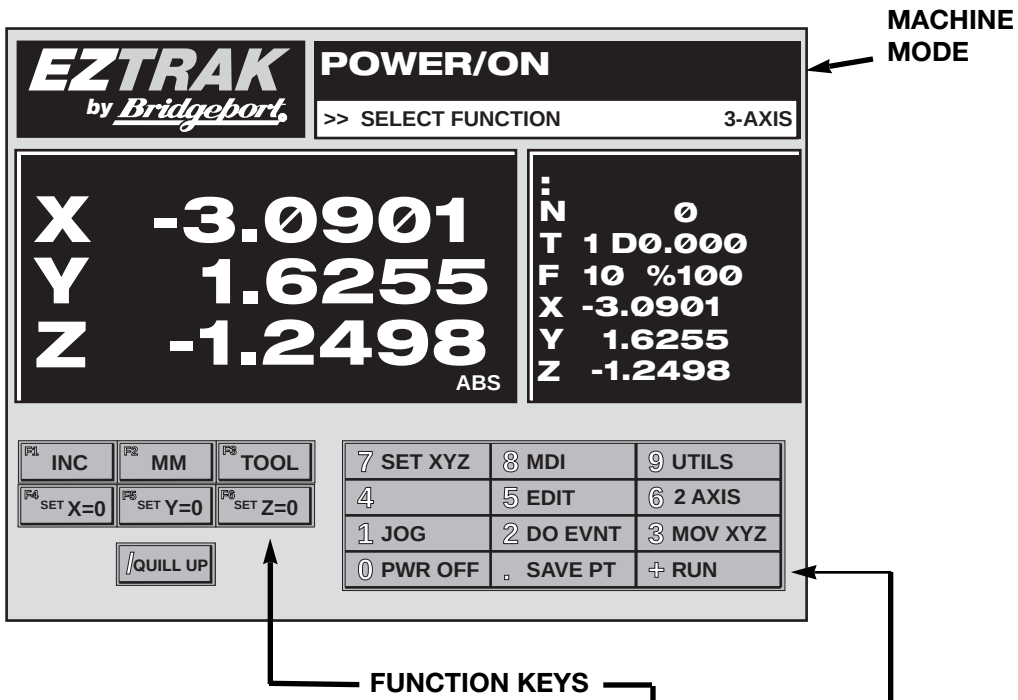
X The X coordinate of the target point (the point the machine is moving towards) is shown here.

Y The Y coordinate of the target point is shown here.

Z The Z coordinate shown here gives the tool depth required at the target point. When the machine is in 2 axis mode and a change in cutting depth is required, the operator is prompted to adjust the Z axis, and the programmed depth is shown here. The Z axis should be adjusted until the large Z coordinate is shown the same as shown here.

At the top of the screen, next to the EZTRAK® logo, the power condition of the axis motor drives is shown in large letters. **POWER/ON** means that the axis drives are currently under power and cannot be adjusted by turning the handwheels. The axes can be jogged using the **JOG X** and **JOG Y** commands from the Jog Screen, or moved by means of the **MOVE ABS** or **MOVE INC** commands. When the **POWER/OFF** condition is shown, then the axes can be moved with the handwheels.

NOTE: POWER/OFF does not mean that power to the spindle or control computer is off!



The EZTRAK® defaults to the POWER/OFF (Manual Mode) condition when the power switch at the back of the machine is first turned on.

Directly to the right of the Axis Power Condition is the Operational Mode display. In Figure 3-3, the **MANUAL Mode** is shown. In the DO EVENT Mode this area displays **DO EVENT** to indicate that the machine is ready for the operator's command. When a program is running in Automatic (Run Continuous) mode, this display shows **AUTO RUN**. When a program is running in Single Block mode, this display reads **BLOCK.**

Figure 3-4 Softkey Display and Control Panel

When a program is loaded, this display reads **SET RUN** to indicate that the EZTRAK is in Run Mode; the new program may be run by pressing 1AUTO or 2BLOCK and then + START. When this display reads **AUTO RDY,** it means the EZTRAK is in the Automatic (Run Continuous) mode, and the program may be run by simply pressing the + START key.

The lower half of the display corresponds to the operator's keyboard and key functions. These are considered '*softkeys*' because the functions of the keys change when the machine mode changes. It is important to pay close attention to the functions of the keys as they are shown on the screen.

3.3 BASIC OPERATION KEY FUNCTIONS

The key functions shown in the BASIC OPERATIONS screen are:

0 PWR OFF

This will remove power from the drives to enable manual operation. In MANUAL mode the readout will display position to the nearest .0005". In the POWER ON mode the large XYZ readout will display actual position and the small XYZ readout will display the target position. When **[0 PWR OFF]** is pressed, a dialogue box will appear telling the operator to jog the Z axis to its farthest minus travel limit.

1 JOG

This command is used to move the X, Y and Z axes, by turning the **JOG +/- SELECTOR** or by using the **AXIS MOTION HANDWHEEL.** Selecting **1 JOG** will turn POWER ON if it is OFF, and call up the JOG SCREEN. From the JOG SCREEN the operator can jog any of the three axes. Use this command (instead of the MOVE commands) to move the X axis, Y axis and Z axis separately, or if the coordinates of the desired position are not known (See MOV XYZ below.)

2 DO EVENT

This command calls up the DO EVENT screen which allows the machine to execute programmed instructions one at a time. This command will also turn POWER ON if it is OFF. (See Chapter 6 , DO EVENT.)

3 MOV XYZ

This command moves the tool to a new point. The coordinates of the new point

must be entered on the screen before the tool is moved. The command will also turn POWER ON if it is OFF. Use MOV XYZ instead of the JOG command to move the X, Y, and Z axes at the same time to a position where the X, Y, and Z coordinates are known. NOTE: If the Z axis is to move in the *positive* direction, it moves first to its target coordinate, then the X and Y axes move to their target coordinates. If the Z axis is to move in the *negative* direction, X and Y move to their target coordinates, then the Z axis moves to its target coordinate.

The **MOV XYZ** command is also used to **HOME THE AXES**. Also, this command allows the operator to move the axes to a set clearance point by pressing the **F1** key.

The X,Y,Z coordinate values can be entered directly, or use **ANGL** or **CALC** subcommands to calculate the X,Y,Z coordinates.

CALC The **CALC** command can be used to enter and evaluate mathematical equations so that points can be calculated. Trigonometric functions, square roots, and exponential functions can be used in the entered equations. See Appendix C for more information on using the calculator.

ANGL

Using the **ANGL** command, the system will convert entered **polar** coordinates into XY coordinates. When the **/ANGL** key is pressed, the screen prompts the user for the following values.

R radius This is the radius value. A positive value must be entered.

A angle This is the angle value in degrees. Either a positive or negative value can be entered.

Z ABS This is the Z coordinate endpoint of the move.

X center This is the X coordinate of the pole.

Y center This is the Y coordinate of the pole.

After all of the fields are filled, press Enter. The system will calculate and then fill in the transformed X and Y values.

5 EDIT.

This command enables the operator to edit a previously created PGM. See chapter 8 for descriptions of how the editor is used.

6 2-AXIS / 3-AXIS

This key will toggle the machine axis control between 2 and 3-axis modes. When this key is pressed, a message will appear that asks the operator to tighten or loosen the Z axis locking mechanism, depending on which axis control mode is desired. (See Chapter 2.)

7 SET XYZ.

This key enables the operator to set the PART PROGRAM coordinate system anywhere, regardless of the current position of the table. The operator enters the values to be set as the PART PROGRAM coordinates for the current table position. For example, if the axes have just been homed and the PART PROGRAM 0,0 is the center of travel, you would enter X 15. Y 6. This sets the current position as 15,6. Entering a Z coordinate will set the Z display to the entered value and set the tool length offset of the active tool to the sum of the entered value and the distance from the Z home position.

> **NOTE: The HOME position is the extreme +X,+Y,+Z machine travel position as set by the combination of a travel limit switch and a marker on the motor feedback encoders.**

This command is also used to set the clearance point on the machine that is used in the **3 MOV XYZ** command. The Clearance Point can be set by pressing **F1** from the **SET XYZ** dialogue box.

F2 - F6 are WORKSHIFTS 1 through 5. Using workshifts, EZTRAK can mill up to five parts in the same work cycle. Keys F2-F6 are used to enter coordinates for each workpiece. See Chapter 7, MDI Programming.

The **CALC** command can also be used in the **SET XYZ** command.
/ **ACT WS (Activate Workshift)**, enables workshifts to begin.

8 MDI

provides the ability to create PART PROGRAMs. See Chapter 7, MDI PROGRAMMING.

9 UTILS

The **9** key in the BASIC OPERATIONS screen temporarily exits the EZTRAK environment to allow the user to run several disk utility programs which are used to copy programs, copy disks, and manage files. A utility is also provided for sending and receiving files from a remote computer. For more information on the File Utilities, see Chapter 10.

F1 ABS-INC.

This changes the context of the XYZ readout from **incremental** to **absolute**. The current mode is indicated at the lower right corner of the XYZ readout with an abbreviation either **ABS** or **INC**. For more information on ABS or INC, see Chapter 4.

F2 INCH-MM.

This switches the system from INCH to METRIC. The current measurement mode is indicated by the position of the decimal point in the XYZ display. If three decimal places are shown, the mode is metric. Four decimal places are shown in inch mode. The EZTRAK® has the ability to save the work mode when the machine is shut down. If the machine is shut down in the INCH mode, it will start up in the INCH mode, if it is shut down in the METRIC mode it will start up in the METRIC mode.

F3 TOOL.

This command allows the operator to select a tool number from the tool library. The tool library can hold up to 24 tools. You can also create tool offsets here using the TLO = Z command. See Chapter 8 in this manual for more information.

F4 set X=0, F5 set Y=0, F6 set Z=0.

These commands reset the X, Y and Z coordinates to 0. In the Absolute mode (**ABS**), this resets the part program origin. In the incremental mode (**INC**) this sets a local coordinate zero point. The set Z function sets the Z display to zero and sets the tool length offset of the active tool to the distance from the home position.

<.> SAVE PT.

This enables the operator to select (in Manual Mode) up to 100 XYZ points which are stored in the system. The points can then be replayed automatically to replicate the same sequence of moves the operator went through manually to

machine the part. For more information on the SAVE PT command, see Chapter 11 TEACH MODE

+ RUN.

The + key in the BASIC OPERATIONS screen calls the **RUN** mode. For more information on the **RUN** mode see Chapter 9 RUN MODE.

/ QU UP

This command automatically raises the quill (in 3-axis mode only) to its uppermost point.

Chapter 4
MANUAL DRO and JOG MODE

4.1 MANUAL DRO and JOG OPERATION

When power to the axis drives is turned **OFF,** the display shows the **MANUAL** mode in the upper right hand corner of the screen. In this mode, the EZTRAK operates as a sophisticated three- axis manual machine with a Digital Readout (DRO).

To remove power from the axis drives, press the **0 PWR OFF** command in the BASIC OPERATION screen. **[NOTE: POWER is only removed from the axis drives. The EZTRAK, retains all system software, part-programs and position information.]** When the **PWR OFF** key is pressed while in 3-axis mode, a message is displayed asking the operator to disengage the Z axis; the power is removed from the axis drives and the machine is set in 2-axis mode. When the PWR OFF key is pressed while in 2-axis mode, no messages are displayed, and power is removed from the axis drives.

There are two modes of MANUAL operation:

1. **ABS.** In the ABSOLUTE mode the XYZ readout displays the distance from the part program origin, i.e., X,Y, Z 0. To set the part program origin press the **7 SET XYZ** key. To select the ABSOLUTE mode use the **F1** key that toggles between ABSOLUTE and INCREMENTAL mode. Selecting the ABSOLUTE mode will restore the previous SET value for the part program origin.

2. **INC**. In the INCREMENTAL mode the XYZ readout will display the distance from the last local setpoint. To SET a local setpoint use the **F4 SET X = 0** command, the **F5 SET Y = 0** command or the **F6 SET Z = 0** command. The designated incremental position register will be zeroed. The local incremental position register will also be zeroed when the **SET XYZ** command is used.

The **ABS** or **INC** mode is indicated by the three letter abbreviation shown below and to the right of the large XYZ coordinate display on the screen.

IN to MM or MM to IN

Use the **F2 MM / IN** key to toggle between inch and metric. The position of the decimal point (4 places for inch, three places for metric) indicates the active system (Inch or Metric).

4.2 THE SAVE POINTS COMMAND

While the EZTRAK® is in the **MANUAL** mode, and a part is being cut by hand, the **SAVE POINTS** command can be used to "**TEACH**" the EZTRAK® to cut the same part.

At each significant point in the manual machining operation, a point can be saved as a **MILL EVENT** or as a **DRILL EVENT**. The EZTRAK records each point and the event type. Up to 100 points can be stored. At the end of the machining process, the points can be replayed, to duplicate the manual machining operation. The file in which the points are stored, named TEACH.PGM, can also be edited in case errors are made, or in case changes are needed.

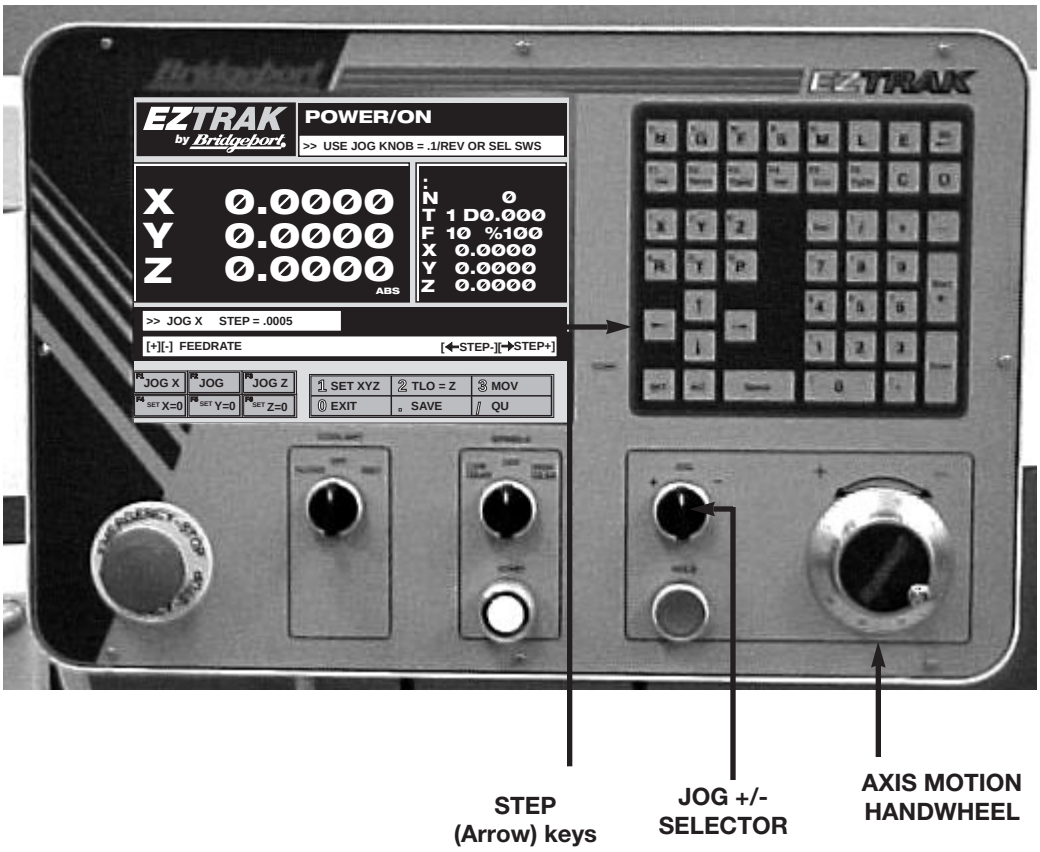
See Chapter 11 in this manual for more information on using the **TEACH** mode.

4.3 THE JOG COMMAND

From the BASIC OPERATIONS screen, the **JOG** menu can be called by pressing the **1 (JOG)** key.

In the JOG menu, each axis can be selected and moved separately by using one of three controls on the control panel. These are: the **JOG+/- SELECTOR,** the rotary **AXIS MOTION SWITCH,** and the **STEP (Arrow)** keys on the keyboard: ← is minus (–) , and → is plus (+).

F1 JOG X is the default, so that the X axis can be jogged. Turn the Jog Knob so that the arrow points toward the "–" sign to move in the minus direction; turn the knob to " + " to jog the X axis in the positive direction.



The **+** and **–** keys on the keyboard are **FEED OVERRIDE** keys. They are used to change the rate at which the table moves. Feed Override can be used to slow the feed rate to zero using the minus (-) key, or increase the feed rate up to 150% using the plus (+) key.

Press **F2 JOG Y** on the keyboard. Using the Jog Knob, jog the table several inches in the – direction. The jog direction can be changed by turning the Jog Knob towards the + sign. Jog the Y axis back towards the center position.

Now try jogging the Y axis by using the **AXIS MOTION WHEEL.** This rotary switch moves the axis 0.125”(3.175 mm) for every full rotation (360º) of the wheel.

If you have a 3-axis EZTRAK, press the **F3 JOG Z** key on the keyboard. to jog the Z axis using the same methods above. (If the machine is in 2 axis mode, this key will be blank.)

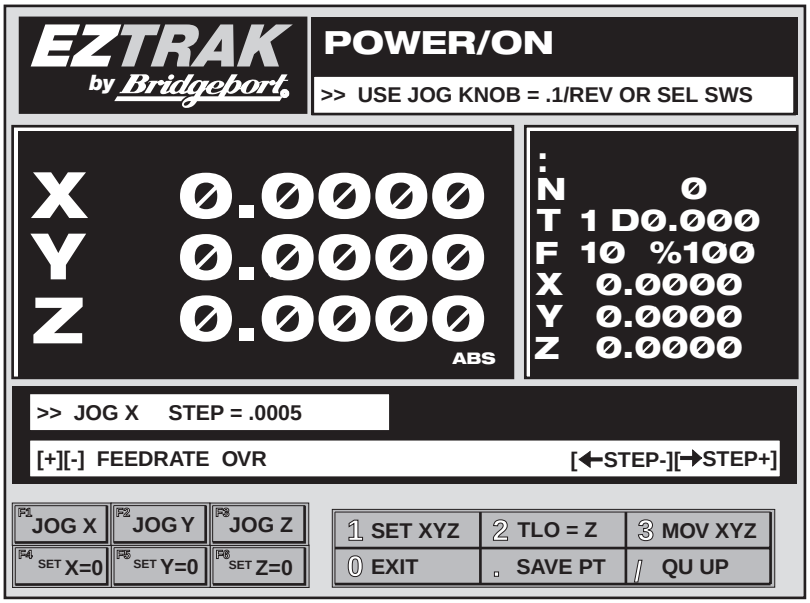


Figure 4-1
JOG SCREEN

[JOG Z], [TLO=Z], and [/QU UP] do not apply to 2-axis machines.

Also try pressing the STEP + and STEP – keys, which are the arrow keys on the control panel **(← is – , and → is +).** These keys jog the axes approximately 0.0005" each time the key is pressed.

The **JOG** command can be used to move the table to a designated position, to continuously move the table at the set feedrate if the **JOG KNOB** is turned or, to move the table in small increments using the AXIS MOTION HANDWHEEL. Use the **1 JOG** command in the BASIC OPERATIONS screen to select the JOG MODE.

The JOG command will turn POWER to the drives ON if it is OFF, call up the HOMING routine if the system has not been HOMED and then call up the JOG screen.

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

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

F1 JOG X

Pressing this key will instruct the machine to JOG the X axis with the JOG BUTTON or the JOG KNOB.

F2 JOG Y

Pressing this key will instruct the machine to JOG the Y axis with the JOG BUTTON or the JOG KNOB.

F3 JOG Z

Pressing this key will instruct the machine to JOG the Z axis with the JOG BUTTON or the JOG KNOB. (3-axis mode only)

F4 SET X=0

Pressing this key in the **JOG** mode resets the X coordinate to zero.

F5 SET Y=0

Pressing this key in the **JOG** mode resets the Y coordinate to zero.

F6 SET Z=0

Pressing this key in the **JOG** mode resets the Z coordinate to zero and sets the tool length offset of the active tool to the distance from the Z home position.

0 EXIT

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

1 SET XYZ

The **SET XYZ** command in the **JOG** mode works the same as the **SET XYZ** command in the BASIC OPERATIONS mode. It prompts the user to reset any, or all, of the X, Y, or Z coordinates.

2 TLO = Z

This command allows the operator to transfer the current Z position value to a selected Tool Length Offset in the Tool Library. If the operator enters a tool number, the actual Z value will then equal the TLO for the tool specified (3-axis mode only). Pressing this command will also prompt the operator to enter a tool diameter that is associated with the TLO.

3 MOV XYZ

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

. SAVE PT

The **SAVE PT** command in the **JOG** mode works the same as the **SAVE PT** command in the BASIC OPERATIONS mode. It allows the operator to store up to 100 points and replay these as positioning or milling events. For more information about the **SAVE PT** command, see Chapter 11 TEACH MODE in this manual.

/ QUILL UP

The / key in the **JOG** mode is labeled **QU UP**. This key moves the quill to the top most position when the operator presses the key. (3-axis mode only)

+ and –

The + key and the – key in the **JOG** mode are used to change the feedrate override. The + key increases the feedrate override value, and the – key decreases the feedrate override value.

4.4 WORKSHIFTS

EZTRAK has the capability to execute up to five workshifts, meaning that you can program the EZTRAK to machine as many as six parts during the same work cycle.

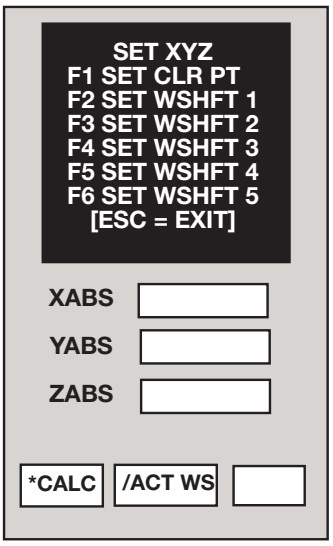


Figure 4-2: Accessed from Basic or Jog Screen by pressing the SETXYZ key.

Setting Workshifts

To program workshifts, you must establish the work coordinates for each part to be milled. The available work coordinate systems in EZTRAK are G54 through G59, in which G54 is the Base or default coordinate system.

Set work coordinates by using the procedure in the following example. Assuming that you have two workpieces clamped to the table, the workpiece on the left is Part 1, and the workpiece on the right is Part 2; assume also that the tool in the spindle has a one-inch endmill, and that the XY zero position is the lower left corner of the part and Z zero is the top..

For Part 1:

(X axis):

1. In the Basic Menu, select 1 JOG.
2. Position the tool to the left of Part 1, then use the handwheel to jog the X axis until the tool touches the side of the workpiece.
3. Select 1 SET XYZ.
4. Select F2 SET WKSHFT 1. Cursor down to the box for X axis, enter –.5 (the radius of the 1-inch endmill), then press ENTER three times to save the X coordinate.

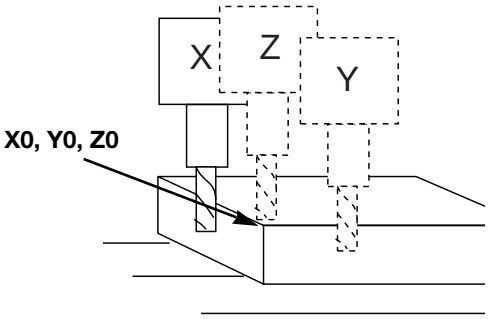


FIGURE 4-3: Touch off three sides of the workpiece with the tool to establish work coordinates.

(Y axis):

1. Select F2 JOG Y. Position the tool to the front of Part 1, then use the handwheel to jog the Y axis until the tool touches the front of the workpiece.
2. Select 1 SET XYZ.
3. Select F2 SET WKSHFT 1. Cursor down to the box for Y axis, enter –.5, then press ENTER twice to save the Y coordinate.

(Z axis):

1. Select F3 JOG Z. Position the tool over Part 1, then use the handwheel to jog the Z axis down until the tip of the tool touches the top of the workpiece.
2. Select 1 SET XYZ.
3. Select F2 SET WKSHFT 1. Cursor down to the box for Z axis and input “0”, then press ENTER.to save the Z coordinate.

The EZTRAK now knows the XYZ work coordinates of the first part to be milled.

For Part 2:

Follow the same steps as for Part 1, but when entering coordinates in SET XYZ, be sure to select F3 SET WKSHFT 2.

To Clear Workshifts

There are two ways to delete workshifts: In EDIT, select **[*TOOLS]** to call up the Workshift and Tool Table. Go down the list and zero out all the workshifts individually.

Alternatively, you may select CLEAR ALL to delete all the workshifts at once, but be forewarned: the CLEAR ALL command will delete all your tool offsets as well.

Make sure you press the (+) key (SAVE) before you ESCAPE, or your deletions will not be saved.

To Activate Workshifts

Workshifts can be activated from the Main Menu, from the Jog Menu, or from a part program. Once activated, they will stay active until another workshift is activated or the operator selects Reset Program. Reset Program activates the default powerup workshift (WORKSHIFT 0).

From the Main Menu:

1. Select 7 SET XYZ
2. Select /ACT WS
3. Enter the desired workshift 0-5.
4. The main DRO will change to the active workshift.

From the Jog Menu:

1. In the Main Menu, select 1 JOG.
2. Select 1 SET XYZ.
3. Select /ACT WS.
4. Enter the desired workshift 0-5.
5. The Main DRO will change to the active workshift.

From a Part Program:

See the Chapter 7 MDI Programming.

Chapter 5
DO EVENT OPERATION

INTRODUCTION

In **DO EVENT** mode, instructions can be programmed and executed one at a time. This means that operations such as drilling bolt circles or milling rectangular pockets can be executed with a single programmed canned cycle instead of being performed manually. All of the canned cycles described in this chapter can be executed from DO EVENT mode.

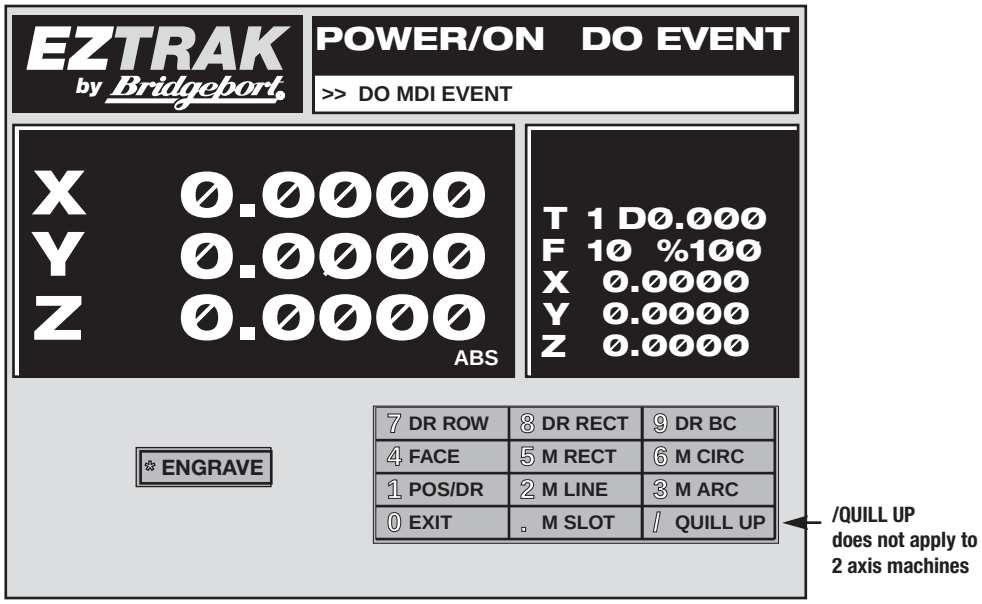


Figure 5-1. DO EVENT Screen

NOTE: DO EVENT operations are run directly from the DO EVENT screen and are not stored in system memory.

IMPORTANT: In the DO EVENT mode, EZTRAK monitors the spindle. If the spindle is OFF when you press the + key to execute a programmed instruction, a message is displayed telling you that the instruction cannot occur unless the spindle is turned ON.

5.1 USING DO EVENT DATA WINDOWS

The DO EVENT commands require the operator to enter data from the keyboard in order to perform the selected functions. Each command in the DO EVENT mode displays a data window at the right hand side of the screen when selected. Each data window is different, and each command requires different data. This chapter describes each of the DO EVENT commands in detail and explains the data required for each command.

Figure 5-4 shows the data window from the **M ARC** command. The data required for this command includes the endpoint location for the tool, the center point of the arc, or its radius value and the milling feed rate. The direction (clockwise or counter-clockwise) is also required in this command.

At the bottom of the data window are three keys, labeled **/GEOM**, ***CALC**, and **+ INC** (this key does not appear in all of the data windows; it appears as **ESC** in several commands). These keys are used to select the GEOMETRY HELP menu, the CALCULATOR functions, and to select the INCREMENTAL data entry mode if it is available.

The **ESC** key can always be used to exit from the selected command. The GEOMETRY HELP and CALCULATOR modes are described briefly below. For more information on incremental programming, see Appendix A in this manual.

F1 XZ PLANE and F3 YZ PLANE do not apply to 2 axis machines

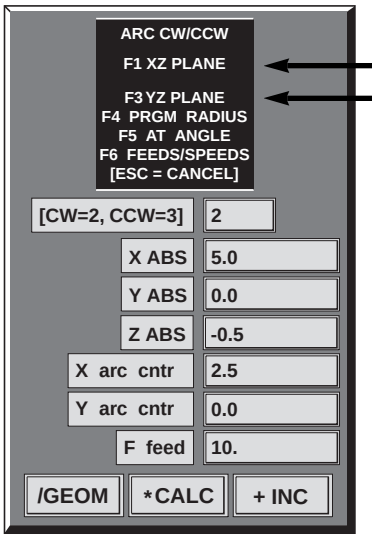


Figure 5-2

/GEOM (GEOMETRY HELP)

The **/GEOM** key selects the GEOMETRY HELP menu. The functions in this menu are used to calculate point locations based on data which might be entered from a blueprint or part drawing. The calculated point coordinates are entered automatically into the DO EVENT command when the calculation is finished. For a complete description of the GEOMETRY HELP functions, see Appendix D in this manual.

*CALC (CALCULATOR)

The ***CALC** key selects the CALCULATOR mode, which can be used to enter and solve a mathematical equation to find a coordinate, or other numeric value. For a complete description of the CALCULATOR mode, see Appendix C in this manual.

FEEDS / SPEEDS

In the milling commands in the DO EVENT and MDI modes, the **F6** key can be used to display a screen which will calculate an appropriate spindle speed and feedrate for the current command. Pertinent data such as work material, tool type and size, and type of cut are required to complete these calculations. See Appendix E in this manual for more information on the FEEDS & SPEEDS calculator.

1 POS/DR

The **POS/DR** screen can be used to program two commands, either **Position** (**Rapid**) or **Drill**. Both of these commands move the tool from its current location to the programmed position. The **Drill** command drills a hole to the programmed depth (3 axis only), once the tool has been moved to the location. If the machine is running in 2-axis mode, the operator will be prompted to drill the hole to the proper depth.

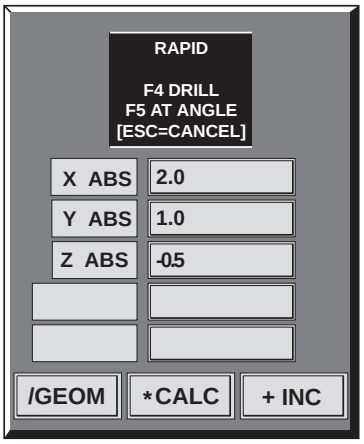


Figure 5-3

For example, the programmed line:

```
0120 DR/PT ABS X2. Y1. Z0.1 Z0.5 Z.05
     Z.05 F10
```

will instruct the EZTRAK to move from the current position to the location (X2, Y1, Z0.1). A hole is then drilled at this location to the depth of **-0.5**.

The **Rapid** command moves the tool to the programmed position then waits for the next command.

> **NOTE: When using the Rapid positioning command in 2-axis mode, it is the operator's responsibility to move the Z axis to a height that clears the workpiece, before the move is executed.**

Both **Rapid** (Figure 5-5) and **Drill** commands can be programmed using **Polar Coordinates**. Pressing the **F5 AT ANGLE** key while the **POS/DR** instruction window is displayed changes the command to use **Polar Coordinates** to enter the new tool position.

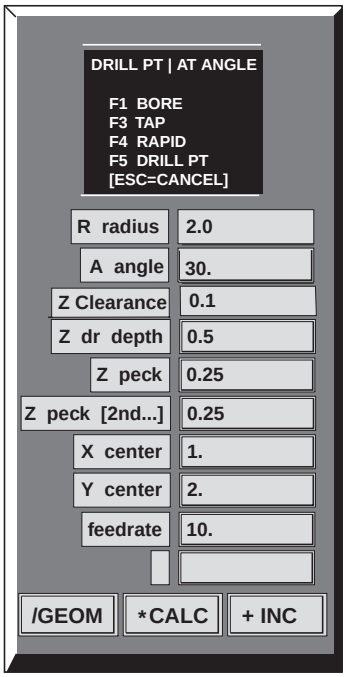


Figure 5-4

For example, the programmed line:

```
0120 DR/ANGL ABS R2. A30 Z0.1 Z0.5 Z0.25
     Z0.25 XC1. YC2. F10.
```

will instruct the EZTRAK® to move the tool to the point shown in Figure 5-7. A hole is then drilled at this location to the depth of **0.5** in 3 axis mode, or the operator is prompted to drill a hole to the depth of **0.5** .in 2 axis mode.

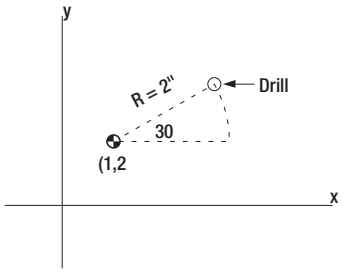


Figure 5-5

When **F4 DRILL** is pressed in the **RAPID** window, a data box like Figure 5-8 appears. This **DRILL PT** window gives the operator a choice of drilling a hole at an angle (F5), boring a hole (F1), tapping a hole (F3), or going back to the **RAPID** window (F4).

DRILL PT

This command instructs the EZTRAK to rapid to X ABS, Y ABS, Z Clearance, feed into the part at the specified feedrate to the incremental Z drill depth, then rapid back up to Z Clearance.

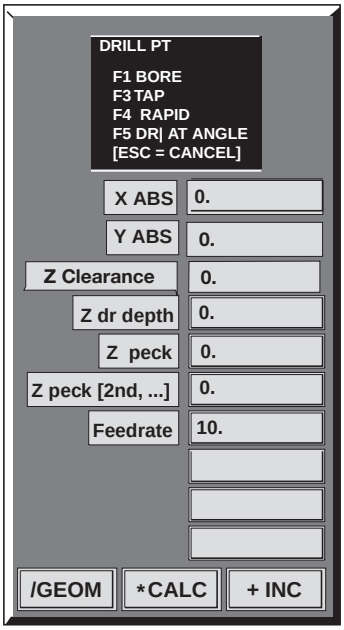


Figure 5-6

X ABS/Y ABS: the X and Y coordinates where the hole is to be drilled.

Z Clearance: the tool clearance above the part

Z dr depth: the unsigned depth (incremental from clearance point) to which the hole is drilled.

Z peck: the first pecking depth (unsigned) for the hole. (If this value equals 0, or more than Z depth, it will be set to **Z depth**).

Z peck [2nd,.]: the second and subsequent pecking depth for the hole. (If this value equals 0, or more than Z depth, it will be set to the **Z peck** value.)

feedrate:: the speed at which the tool moves into the part.

F1 BORE

This command instructs the EZTRAK to rapid to X ABS, Y ABS, Z Clearance, feed into the part at the specified feedrate to the incremental Z Dr Depth, pause for the specified Dwell amount, then feed back up to Z Clearance.

X ABS/Y ABS: the X and Y coordinates where the hole is to be bored.

Z Clearance tool clearance above the part.

Z depth: the depth to which the hole is to be bored.

Feedrate: the speed at which the tool moves into the part.

Dwell: the time (in seconds) the tool should pause at the programmed depth before retracting.

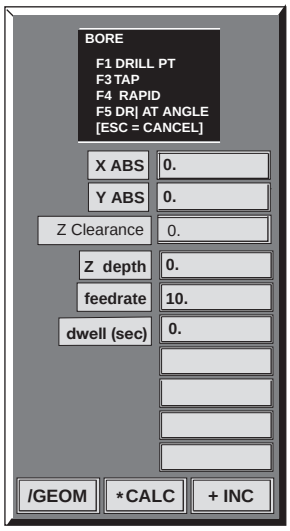


Figure 5-9

F3 TAP

This command instructs the EZTRAK to rapid to X ABS, Y ABS, Z Clearance, feed into the part at the specified feedrate to the incremental Z Dr Depth, pause for the specified Dwell amount, then feed back up to Z Clearance.

X ABS/Y ABS: the X and Y coordinates where the hole is to be tapped.

Z Clearance tool clearance above the part.

Z depth: the depth (incremental from clearance point) to which the hole is tapped.

Feedrate: the speed at which the tool moves into the part.

Dwell: the time (in seconds) the tool should pause at the programmed depth before retracting.

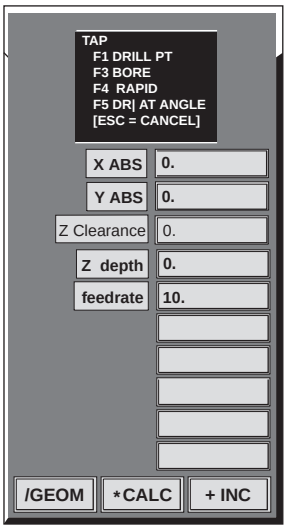


Figure 5-10

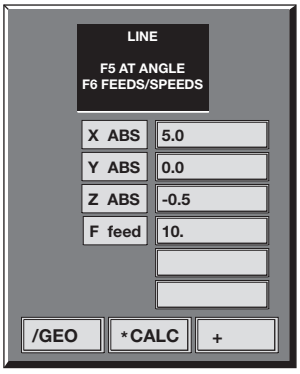


Figure 5-11

2 M LINE

The **M LINE** command causes the tool to move in a straight line at the programmed **Feedrate** to the location specified by the **X**, **Y**, and **Z** coordinates. When the **M LINE** command is selected from the DO EVENT screen, the screen displays the box shown in Figure 5-11. (If the EZTRAK has been set in **incremental mode,** the incremental distances for the line are required.)

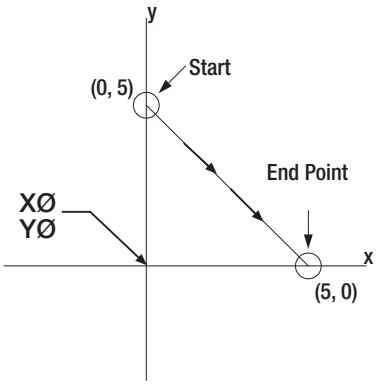


Figure 5-12

X ABS: the X coordinate of the line endpoint.
Y ABS: the Y coordinate of the line endpoint.
Z ABS: the Z coordinate showing the milling depth in 2-axis mode and the Z coordinate of the line endpoint in 3-axis mode.
F feed: the feedrate for the milling operation.

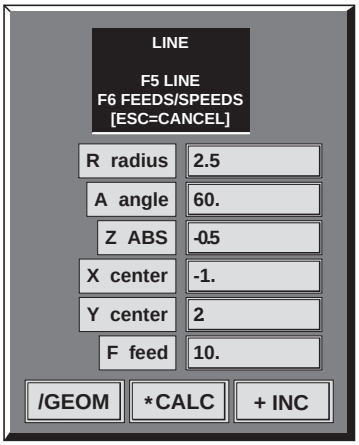


Figure 5-13

For example, the programmed line:

```
0130 LINE ABS X5. Y0. Z-0.5 F10.
```

will instruct the EZTRAK to mill a straight line from the current position to the location (5,0). If the tool was positioned at (0,5) when this line was executed, the tool movement would look like Figure 5-12 shown at right.

The **MLINE** command can be programmed using **Polar Coordinates**. Pressing the **F5 AT ANGLE** key while the **MLINE** instruction window is displayed changes the command to use **Polar Coordinates** to enter the new tool position. For example, if the current tool location is (3,2), the programmed line:

```
0120 LINE/ANGL ABS R2.5 A60 Z-0.5 XC-1.
     YC2. F10.
```

will instruct the tool to move to the point shown in Figure 5-14. The movement of the tool in this command is still a straight line from its previous position; however, the point is defined with radius and angle values.

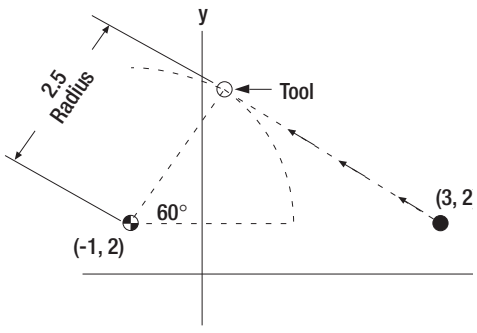


Figure 5-14

3 M ARC

F1 XZ PLANE and F3 YZ PLANE do not apply to 2 axis machines

The **M ARC** command is used to mill a path along an arc. The arc may be specified :

1. by its endpoint and the coordinates of the center of the arc.

2. by its endpoint in polar coordinates and the coordinates of the center of the arc.

3. by its endpoint and the radius of the arc if the arc is 180 degrees or less.

The arc can also be defined in either the XY Plane, the XZ Plane , or the YZ Plane.

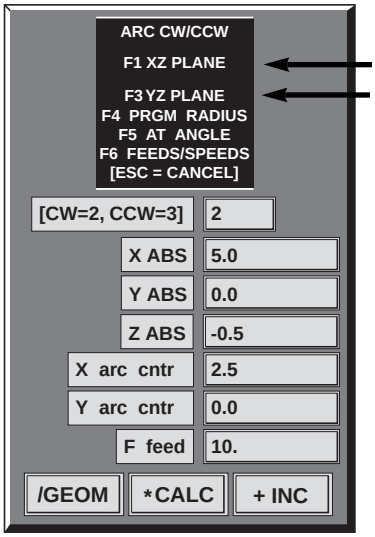


Figure 5-15

When the **M ARC** command is selected from the DO EVENT screen, the screen appears as in Figure 5-15.

CW/CCW: direction that the arc is to be milled (clockwise or counter-clockwise).
X ABS: X coordinate of the endpoint of the arc.
Y ABS: Y coordinate of the endpoint of the arc.
Z ABS: Z coordinate of the endpoint of the arc.
[X arc cntr: X coordinate of the centerpoint of the arc.
Y arc cntr]: Y coordinate of the centerpoint of the arc.
or [R radius]: the Radius of the arc (not necessary if arc cntr is used).
F feed: the feedrate in inches (or mm) per minute of the milling move.

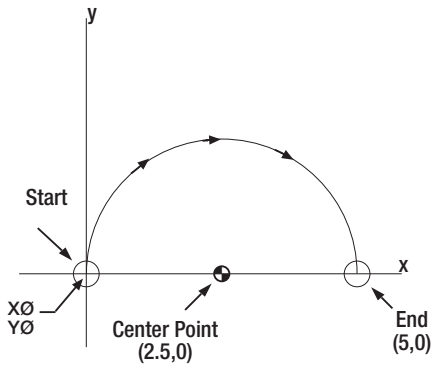


Figure 5-16

For example, the instruction:

```
0150 ARC|CNTRPT ABS CW X5. Y0. Z-0.5
     XC2.5 YC0. F10.
```

will instruct the EZTRAK to mill an arc from the current

position to the location (5,0). In this instruction, the arc is specified by its center point (2.5,0).

If the tool were at the location (0,0) when this instruction was executed, the tool movement would look like Figure 5-16. If this same arc were programmed by the radius of the arc, it would look like this:

```
0150 ARC|RADIUS ABS X5. Y0.
     Z-0.5 R2.5 F10.
```

This line produces the same motion shown in Figure 5-16 for the **ARC|CNTRPT** line.

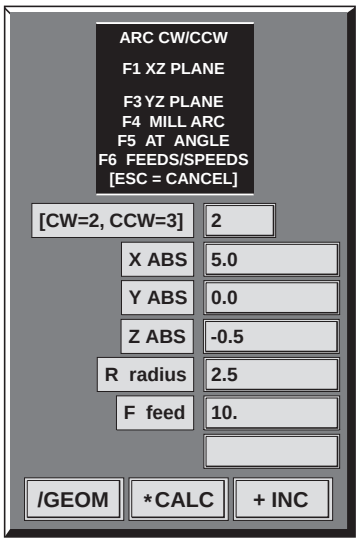


Figure 5-17

An arc can also be programmed by specifying the endpoint using **Polar Coordinates**. Pressing the **F5** key while the **M ARC** command window is displayed changes the command to accept the Radius, Angle, and Arc Center parameters. These specify the endpoint of the M ARC command.

For example: the line programmed as shown in Figure 5-18:

```
0020 ARC|ANGL ARC CW R2.5 A180.
     Z-0.5 XC2.5 YC0 F10.
```

will execute the same movement as shown in Figure 5-16.

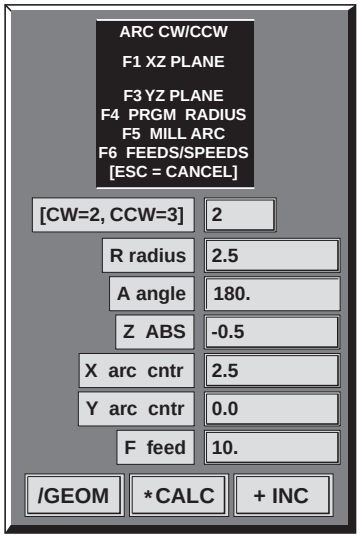


Figure 5-18

NOTE

The angle must be input as an absolute value.

NOTE: If the arc is defined in the XY plane, and the Z ABS differs from the current Z position, a helical move will be performed. You will also get a helical move if the arc is defined in the XZ plane and the Y axis is to be moved, or if the arc is defined in the YZ plane and the X axis is to be moved.

4 FACE

The **FACE** command face-mills an area, designated by its length and width.

X inc dist: the incremental distance to be milled along the X axis.
Y inc dist: the incremental distance to be milled along the Y axis.
Y stepover: the Y axis stepover.
F feed: the feedrate in inches (mm) per minute.

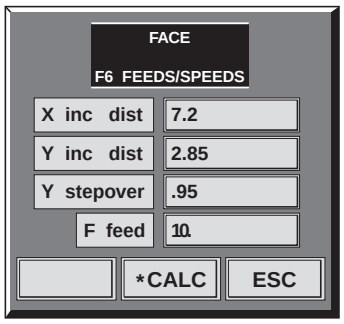


Figure 5-19

For example, the programmed instruction:

```
0160 FACE X7.2 Y2.85 Y.95 F10.
```

will face-mill the 6.0 x 3.75 block shown in Figure 5-20.

The tool starts at the lower left corner of the face area when this operation is executed.

NOTE: Before programming this cycle, position the tool by using the 1 POS/DR command, the 3 MOV XYZ command, or jogging the axes.

NOTE: It is possible to start at the top right hand corner of the face area, and proceed down and to the left. In this case, the X distance and Y distance values are entered as negatives. The Y stepover value is always entered as an unsigned distance.

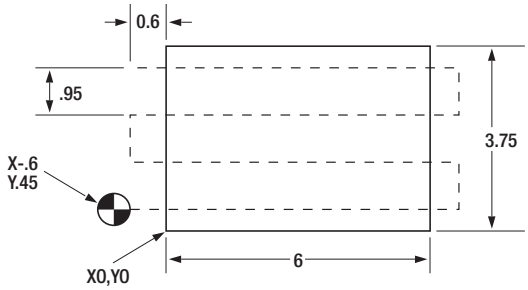


Figure 5-20

5 M RECT

The **M RECT** command is used to call three different milling routines. These routines each cut the rectangle in a different way. The routine called is determined by the value entered into the first field in the **M RECT** window. Enter a **0** to cut the outside border of the rectangle, or a **1** to cut the inside border of the rectangle. Press the **F4** key to mill the interior of the rectangle as a pocket. Press the **F5** key to use the edge reference mode. **Edge Reference** uses the outside, inside and pocket commands.

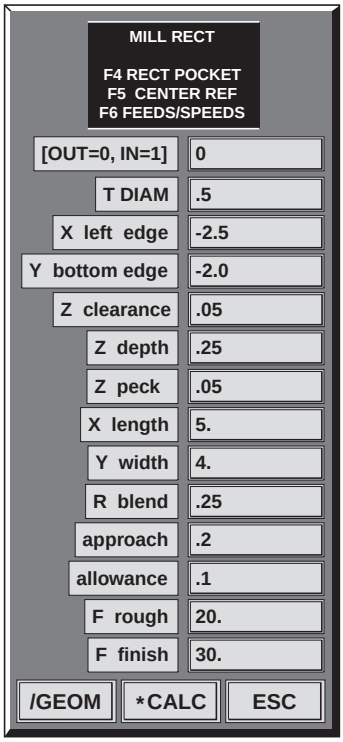


Figure 5-21

The system calculates the starting point for each cycle based on the approach value given. Each of these cycles performs both a roughing pass and a finishing pass.

.

NOTE: Pressing the F5 key changes the M RECT command so that the rectangle is referenced by its left edge, and its bottom edge instead of the center coordinate.

OUT/IN:	determines what part of the rectangle is cut.
T DIAM:	the diameter of the tool to be used.
X cntr Y cntr:	the center point of the rectangle.
Z clearance:	the tool clearance above the part.
Z depth:	the unsigned depth, incremental from clearance.
Z peck :	the peck depth of the cut.
X length:	the unsigned length.
Y width:	the unsigned width.
R blend:	the corner radius.
approach:	distance from the entry point.
allowance :	amount of material left by rough cut to be removed by finish cut.
F rough:	the roughing feed rate.
F finish:	the finishing feed rate.

NOTE: Zero (0) can be entered for approach and/or allowance and/or peck.

OUTSIDE RECT MILL

For example, this instruction:

```
RECT|CNTR OUT X0. Y0. Z.05 Z.25 Z.05
  X5. Y4. R.25 P.2 P.1 D.5 F20. F30.
```

Cuts the path shown in Figure 5-23.

NOTE: In the EDGE REFERENCE mode the example in Figure 5-22 would be programmed as shown in Figure 5-24, creating the instruction:

```
RECT|EDGE OUT X-2.5 Y-2.0 Z.05 Z.25
      Z.05 X5. Y4. R.25 P.2 P.1 D.5
      F20. F30.
```

Z Clearance and Z Peck do not apply to 2 axis machines

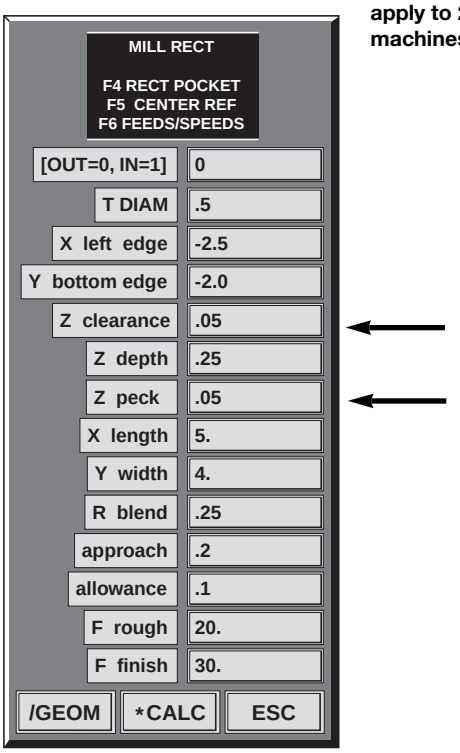


Figure 5-24

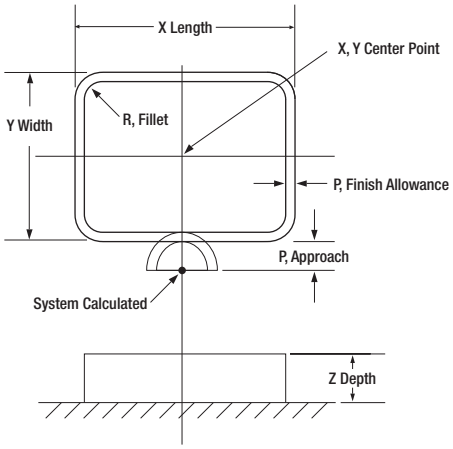


Figure 5-22

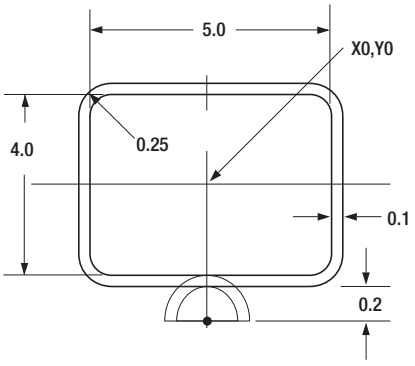


Figure 5-23

INSIDE RECT MILL

Z Clearance and Z Peck do not apply to 2 axis machines

Figure 5-25 programs an inside rectangle. This data creates the instruction below:

```
RECT|CNTR IN X0. Y0. Z.05 Z.25 Z.05 X5.
      Y4. R.25 P.2 P.1 D.5 F20. F30.
```

which cuts the path shown in Figure 5-27.

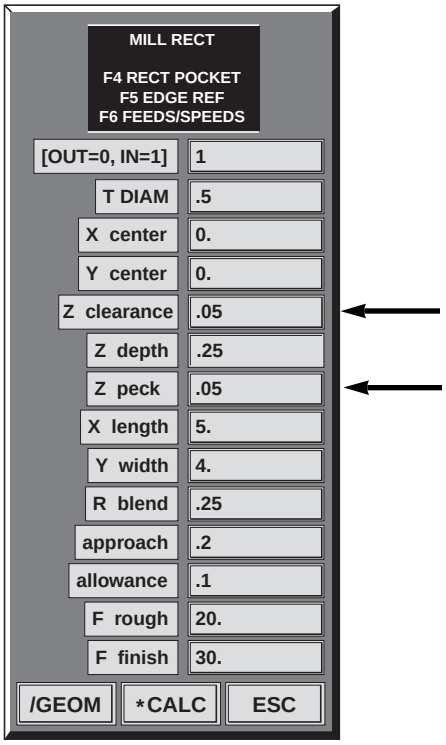


Figure 5-25

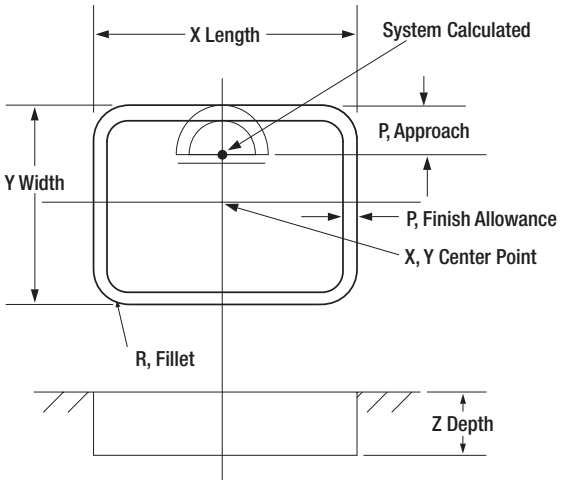


Figure 5-26

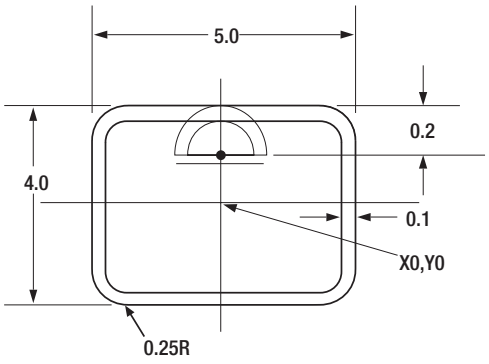


Figure 5-27

POCKET RECT MILL

For example, this instruction:

```
RECT PKT|CNTR X0. Y0. Z0. Z.25 Z.25 X5.
Y4. R.25 P.25 P.2 P.375 D.5 F20. F30.
```

Cuts the path shown in Figure 5-30.

NOTE:
PKT stepover is the tool stepover used in cutting the pocket.

The rectangular pocket may also be referenced by its left edge and bottom edge, instead of the center coordinate. Press the **F5** key to use the **EDGE REFERENCE** mode.

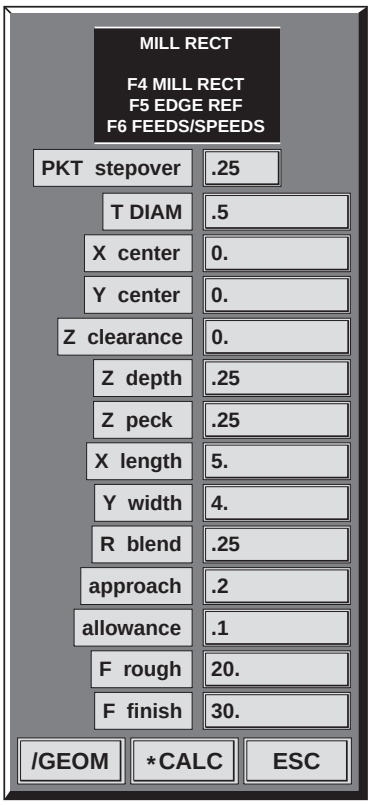


Figure 5-29

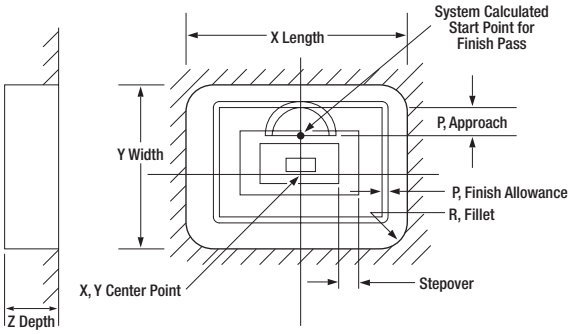


Figure 5-28

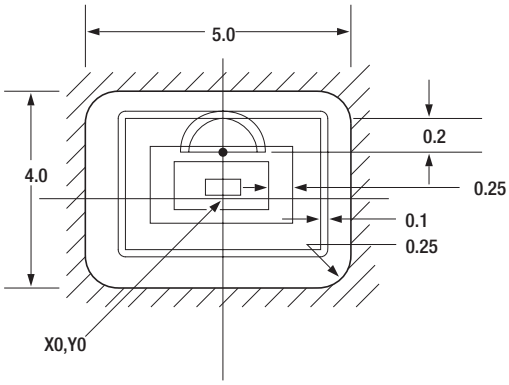


Figure 5-30

6 M CIRC

Z Clearance and Z Depth do not apply to 2 axis machines

The **M CIRC** command is used to call four different milling routines. These routines each cut a different circle. The routine called is determined by the value entered into the first field in the **M CIRC** window. Enter **0** to cut the outside border of a circle. Enter **1** to cut the inside border of a circle. Select **F3** to cut an ellipse. Press the **F4** key to mill a circular pocket.

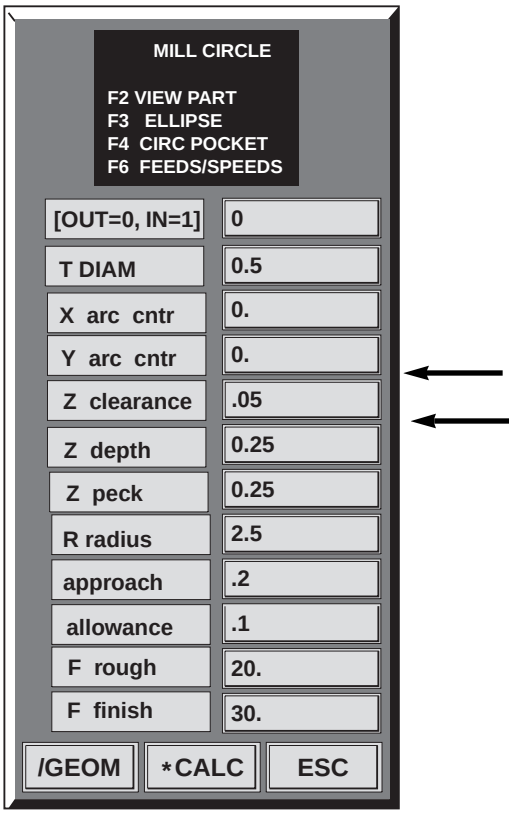


Figure 5-31

The system calculates the starting point for each cycle based on the approach value given. Each of these cycles performs both a roughing pass and a finishing pass..

OUT/IN:	determines what will be cut.
T DIAM:	the diameter of the tool to be used.
X, Y arc cntr	the center point of the circle.
Z clearance:	the height of the tool over the part.
Z depth:	the total depth of the cut (inc. from clear).
Z peck:	the pecking depth.
R radius:	the circle radius.
approach:	the initial entry clearance.
allowance:	the finish cut distance.
F rough/finish	the rough and finish feeds.

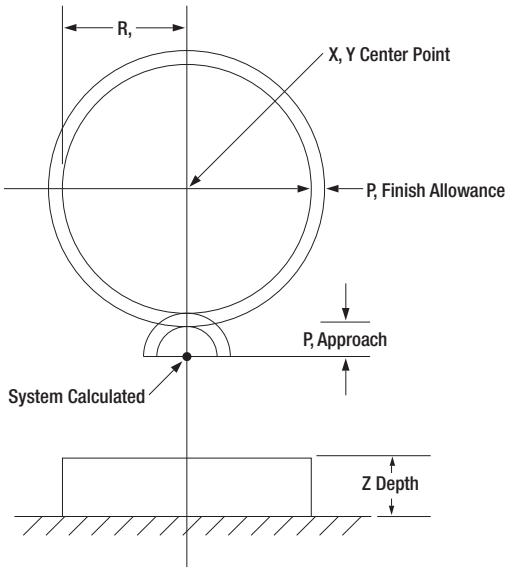


Figure 5-32

OUTSIDE CIRCLE MILL

NOTE: For pockets and inside circles, the tool radius, approach, allowance and stepover added together cannot exceed the radius of the pocket or circle.

For example, this instruction:

```
CIRCLE OUT X0. Y0. Z0. Z.25 Z.25 R2.5
        P.2 P.1 D.5 F20. F30.
```

cuts the path shown in Figure 5-33.

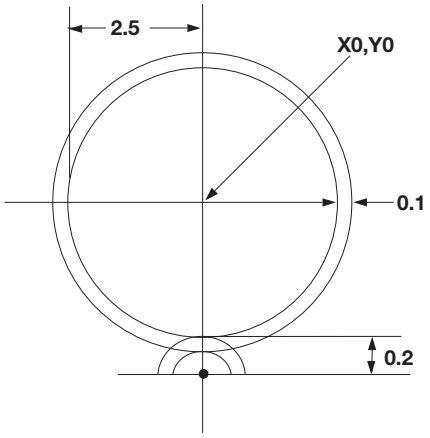


Figure 5-33

INSIDE CIRCLE MILL

For example, this instruction:

```
CIRCLE IN X0. Y0. Z0. Z.25 Z.25 R2.5
        P.2 P.1 D.5 F20. F30.
```

cuts the path shown in Figure 5-35.

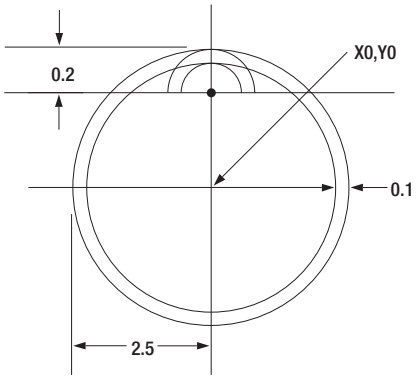


Figure 5-35

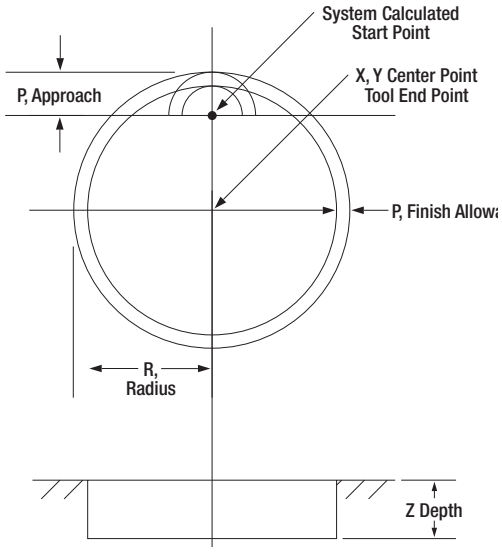


Figure 5-34

F3 ELLIPSE

The F3 ELLIPSE command has three milling routines, each of which cuts a different ellipse. The routine called is determined by the value entered into the first field in the MILL ELLIPSE window. Enter **0** to cut the **outside border** of an ellipse. Enter **1** to cut the **inside border** of an ellipse. Press the **F4** key to **mill the interior** of the ellipse as a pocket.

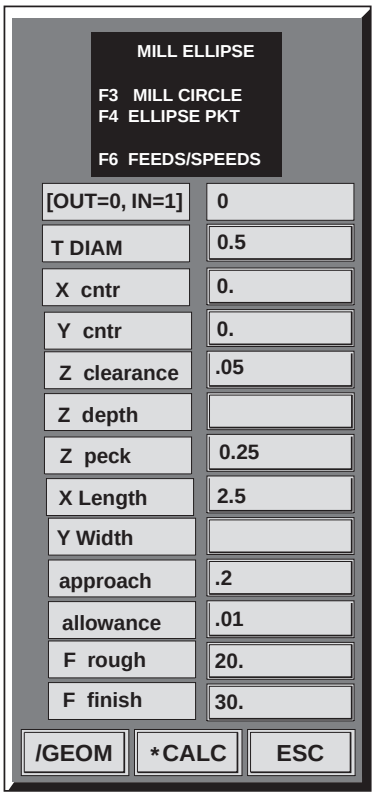


Figure 5-37

The system calculates the starting point for each cycle based on the approach value given. Each of these cycles performs both a roughing pass and a finishing pass. In 2-axis mode, when the cycle is finished, the operator is prompted to raise the tool.

IN/OUT determines what will be cut (outside/inside/profile)
T DIAM the diameter of the tool to be used
X cntr/Y cntr the centerpoint of the ellipse
Z clearance the height of the tool over the part (3 AXIS ONLY)
Z Depth the total depth of the cut (inc. from clear)
Z peck the pecking depth (3 AXIS ONLY)
X Length the X length out to out
Y Length the Y length out to out
Approach the initial entry clearance
Allowance amount of material left by rough cut to be removed by the finish cut
F rough the feed rate for the roughing passes
F finish the feed rate for the finish pass

NOTE: For inside ellipses, the tool radius, approach and allowance added together cannot exceed the minor radius of the ellipse.

NOTE: Zero (0) can be entered for approach and/or allowance and/or peck.

For example, this instruction:

0010 ELLIPSE OUT X0. Y0. Z.1 Z.5 Z.25 X2.
Y1. P.2 P.050 F20. F15.

cuts the path shown in Figure 5-38.

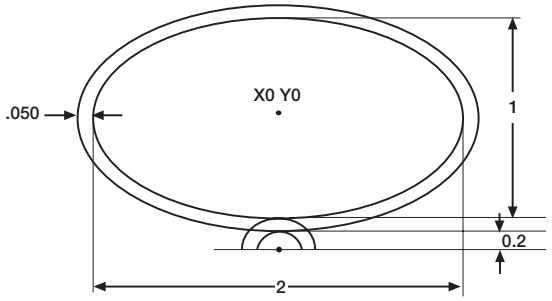
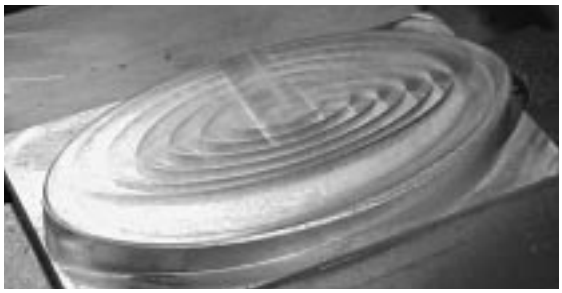


Figure 5-38: Outside Ellipse



In the Ellipse In/Out window, select F4 to mill an **Ellipse Pocket.** The menu which appears lists the same parameters as Figure 5-37 except that the In/Out window is replaced by **PKT Stepover.**

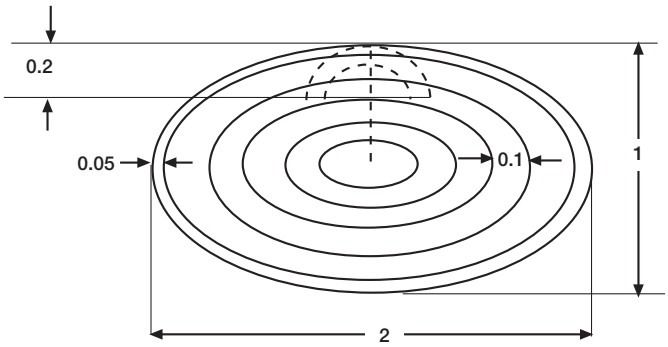
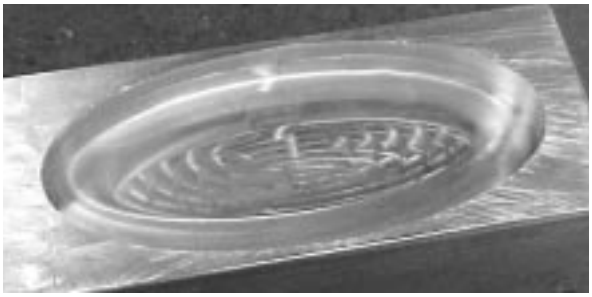


Figure 5-39: Ellipse Pocket



F4 POCKET CIRCLE MILL

For example, this instruction:

```
CIRCLE PKT X0. Y0. Z0. Z.25 Z.25 R2.5
     P.2 P.1 P.1875 D.5 F20. F30.
```

cuts the path shown in Figure 5-40.

NOTE:
The sum of the Stepover and Tool Radius cannot exceed the Pocket Radius.

PKT Stepover is the stepover for clearing out the pocket.

To skip holes in a series, see the Skip Holes function in MDI mode (Chapter 7).

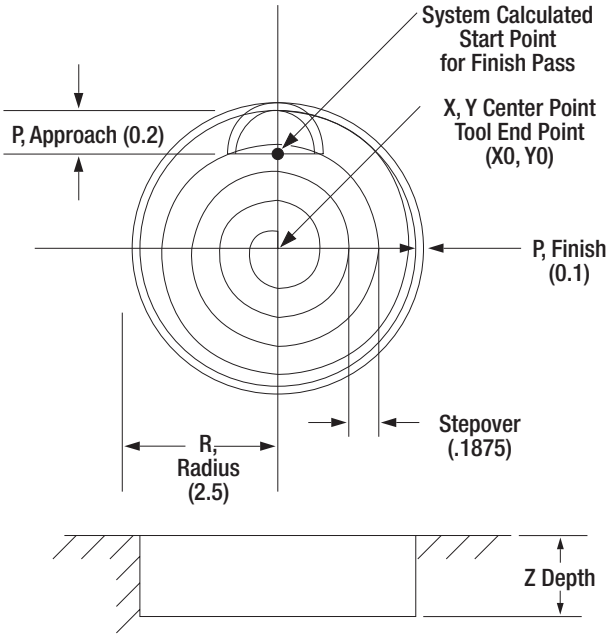


Figure 5-40

7 DR ROW

The **DR ROW** command drills a row of holes beginning at a specified starting point. The holes in the row are evenly spaced from the set starting point over the row length given by the X and Y incremental distance values.

X Y ABS: the start point for the row.
Z clearance: the clearance over the part .
Z depth: the depth (unsigned) (inc from clr pt).
Z peck: the first peck depth.
Z peck [2nd,...]: the second and subsequent peck depths.
X inc dis: the incremental X distance from the center of the first hole to the center of the last hole.
Y inc dis: the incremental Y distance from the center of the first hole to the center of the last hole.
 # holes: the number of holes.
feedrate: the rate the tool is fed into the work.

.

```
0010 DR/ROW ABS X2.5 Y1.5 Z.1 Z1.5
Z.1 Z.1 X3. Y1.5 P4 F4.
```

Z Clearance , Z Peck , and Feedrate do not apply to 2 axis machines

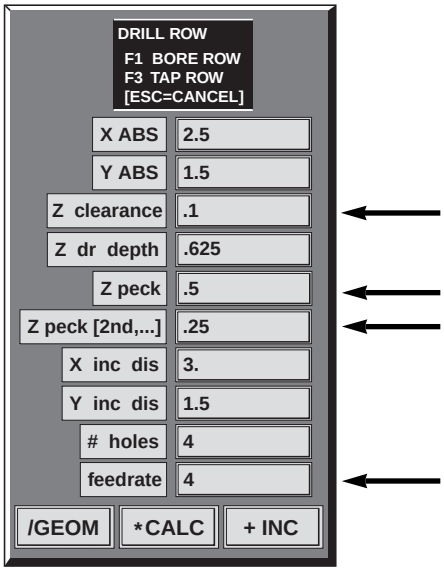


Figure 5-41

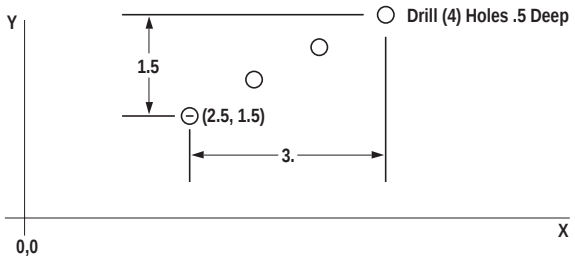


Figure 5-42

8 DR BOX

Z Clearance , Z Peck , and Feedrate do not apply to 2 axis machines

This command will drill holes along the perimeter of a rectangle.

X abs/Y abs: the start point (bottom, left hole).

Z clearance: the clearance of the tool above the part.

Z dr depth: the depth (unsigned) (inc from clr pt).

Z peck: the first peck depth (from clr pt).

Z peck [2nd,...]: the second and subsequent peck depths.

X inc dis: the incremental X distance from the center of the left column of holes to the center of the right column of holes.

Y inc dis: the incremental Y distance from the center of the bottom row of holes to the center of the top row of holes.

 # X / # Y holes: the number of holes in the X direction and the number of holes in the Y direction.

feedrate: the speed at which the tool moves into the part .

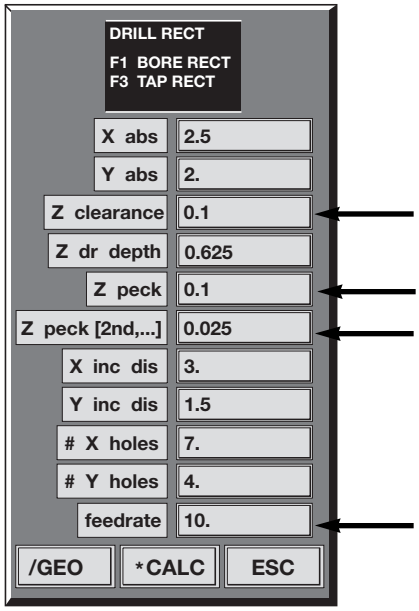


Figure 5-43

For example, the following program will drill the series of holes illustrated in Figure 5-41:

```
0020 DR/RECT X2.5 Y2. Z0.25
Z0.625 Z0.1 Z0.025 X3. Y1.5 P7
P4 F10.
```

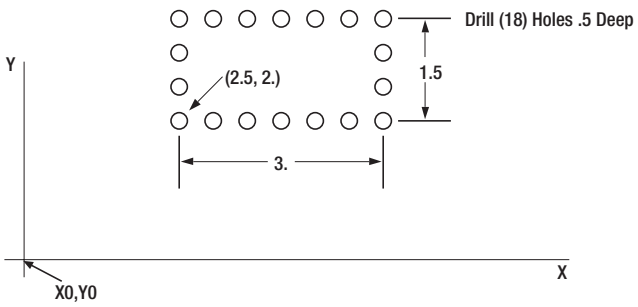


Figure 5-44

To skip holes in a series, see the Skip Holes function in MDI mode (Chapter 7).

9 DR BC

Z Clearance , Z Peck , and Feedrate do not apply to 2 axis machines

```
DR/BC R1.75 Xc2. YC3. Z.25 Z.625 Z.5
    Z.25 A30. P6 F10.
```

This command will drill a series of holes along the perimeter of a bolt circle. The holes are spaced evenly from the start angle around the perimeter of the circle. Pressing the **F4** key changes this command to drill holes along an arc. Figure 5-45 shows an example of the arc command.

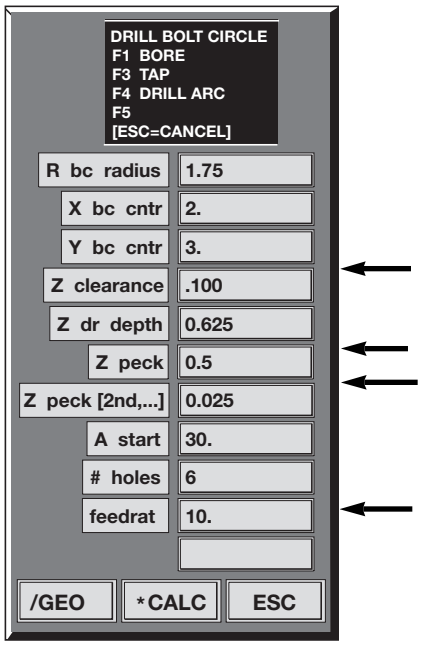


Figure 5-45

R radius: the bolt circle radius.
X cntr/Ycntr: the bolt circle center.
Z clearance: the distance above the part.
Z dr depth: the depth (unsigned) (inc from clr pt).
Z peck: the first peck depth (from clr pt).
Z peck [2nd,...]: the second and subsequent peck depths.
A start: the start angle.
 # holes: the number of holes.
feedrate: the speed the tool moves into the part.

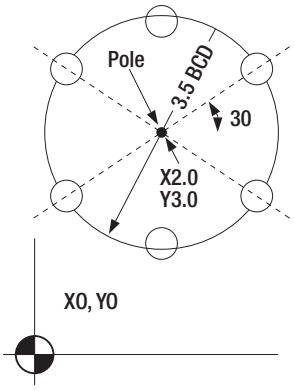


Figure 5-46

F4 DRILL

R bc radius: the bolt circle radius.
X cntr/Y cntr: the bolt circle center.
Z clearance : the clearance height above the part.
Z dr depth: the drill depth (unsigned) (inc from clearance point).
Z peck: the first peck depth (from clearance point).
Z peck [2nd,..]: the second and subsequent peck depths.
A start: the start angle.
A incr dis: the incremental angle from the first hole to the last hole.
 # holes: the number of holes.
feedrate: the speed the tool moves into the part.

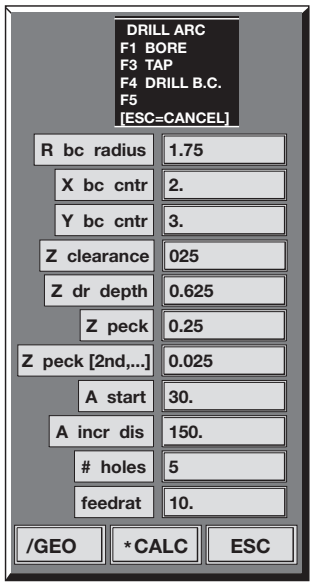


Figure 5-47

For example the programmed instruction:

```
DR|ARC R1.75 XC2.0 YC3.0 Z0.25
Z0.625 Z0.25 Z0.025 A30 A150 P5
F10.
```

To skip holes in a series, see the Skip Holes function in MDI mode (Chapter 7).

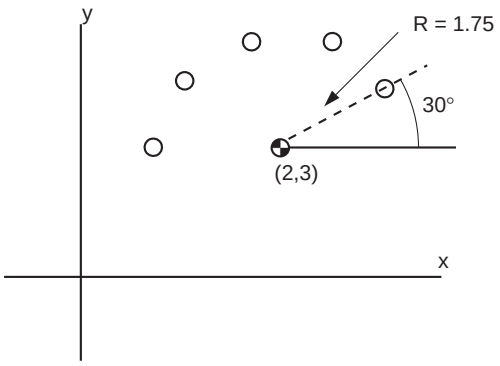


Figure 5-48

Drills a set of 5 holes as shown in Figure 5-45.

. M SLOT

. M SLOT mills a slot at any angle to the X axis. Pressing the F4 key changes this command to mill a n *arc slot*. (See also MSLOT in Chapter 7.)

T DIAM: the tool diameter.
X cntr/Y cntr: the center point of the left slot arc (see Fig. 5-47, below).
Z clearance: the clearance height above the part.
Z depth: the slot depth (inc from clr pt).
Z peck: the pecking depth (from clr pt).
out to out_: the length of the slot end to end (the length must be a positive number).
slot width: the slot diameter.
rotated angle: the angular rotation from the X-axis.

NOTE The SLOT instruction temporarily executes a G97 (coordinate shift) causing the axes display to reset to zero before cutting the slot. The axes are reset to their previous values when the SLOT instruction is completed.

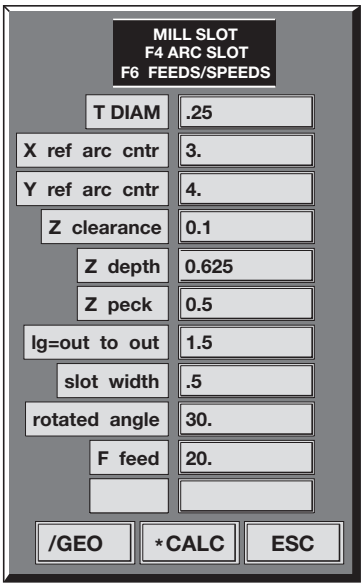


Figure 5-49

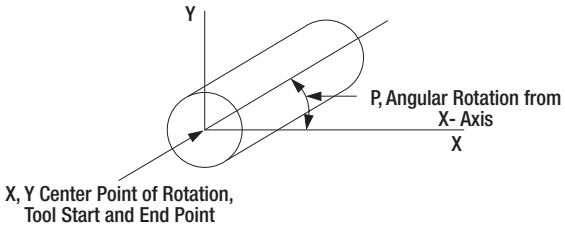


Figure 5-50

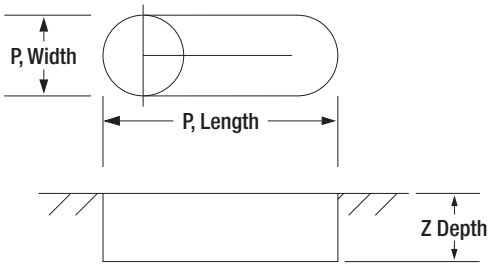


Figure 5-51

* ENGRAVE

The ENGRAVE command cuts a row of text in the form of letters and decimal numbers.

When *ENGRAVE is selected in DO EVENT mode the user is prompted for the engraving parameters (Figure 5-52).

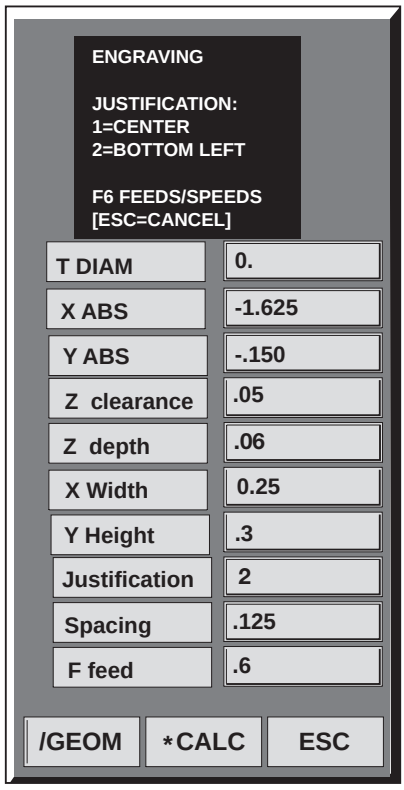


Figure 5-52

T DIAM	the tool diameter
X ABS/Y ABS	the start point for the text
Z clearance	the clearance over the part
Z Depth	the depth (unsigned) (incremental from clearpoint)
X Width	the width of each character
Y Height	the height of each character
Justification (1,2)	the start point location. 1 = start point is located at the center of the first character. 2 = start point is located at the bottom left of the first character.
Spacing	the distance between characters
Feed rate	the rate at which the tool is fed into the work

For example, we'll use the following text:

3 AXIS EZ-TRAK

each character to be 2.5 in. high (Y height) and .25 in wide (X width).

Valid characters are **0-9, A-Z** (upper case only), **+, -**, and **.**.

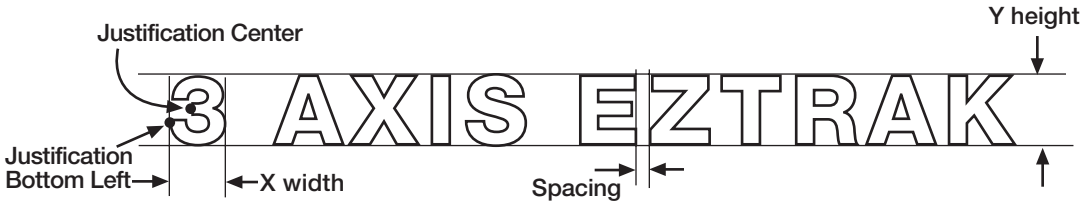


Figure 5-53

After the user has entered in the parameters, a second screen appears, asking for the text to be engraved.

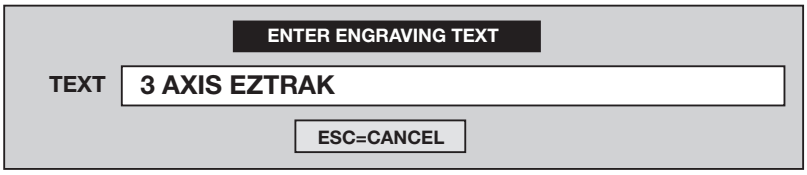


Figure 5-54

NOTE: If the host machine has the new style keyboard the text can be entered directly (Figure 5-54). If it has the old style keyboard, the user must enter *text codes* that correspond to each engraving character. The code list (Figure 5-55) appears after parameters have been entered. Text is typed into the space beside "Codes", where there is room for 50 characters total. Each code number must be separated by a backslash.

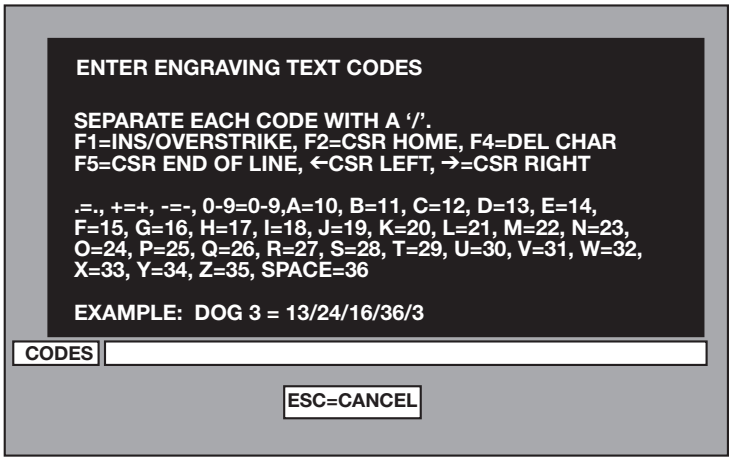


Figure 5-55

After the text is entered, two lines of conversational text appear. The first line consists of the engraving parameters (i.e. tool diameter, spacing, etc.) and the second line contains the actual text to be engraved. (The lines *must* be in that order.) After verifying that the parameters and text are correct, press the **+** key to start the engraving cycle.

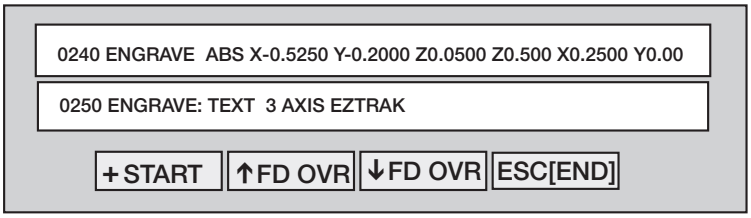


Figure 5-56

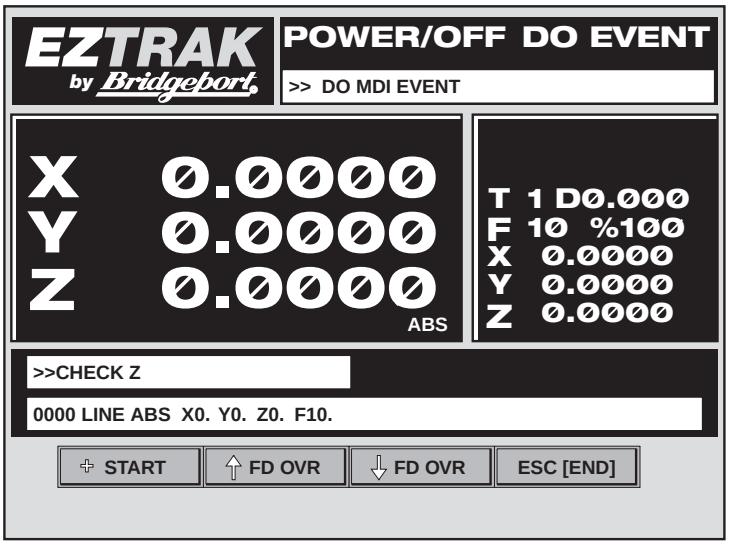


Figure 5-58

5.2 EXECUTING DO EVENT COMMANDS

After the DO EVENT data fields have been filled in, the screen displays a `>.` Below this message is the programmed instruction that is about to be executed. Be sure the spindle is turned on when an instruction has been programmed.

+ START
will begin execution of the programmed operation

↑ ↓ FEED OVERRIDE
increases or decreases the feed override value.

ESC
will abort the operation, and return to the DO EVENT screen. This key can be pressed any time, before or during the programmed cycle execution.
Use CAUTION if this key is pressed while the tool is cutting.

This screen is displayed until the complete cycle is finished. The main DO EVENT screen is then displayed.

To exit DO EVENT mode, select **0 EXIT** in the DO EVENT screen menu.

Chapter 6
EZTRAK TUTORIAL

INTRODUCTION

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

WARNING!! It is extremely important that you do the entire Tutorial from beginning to end; do not attempt to start in the middle, since the instructions in some sections may depend on operations from previous sections. You could crash the part, damaging the machine and/or injuring yourself.

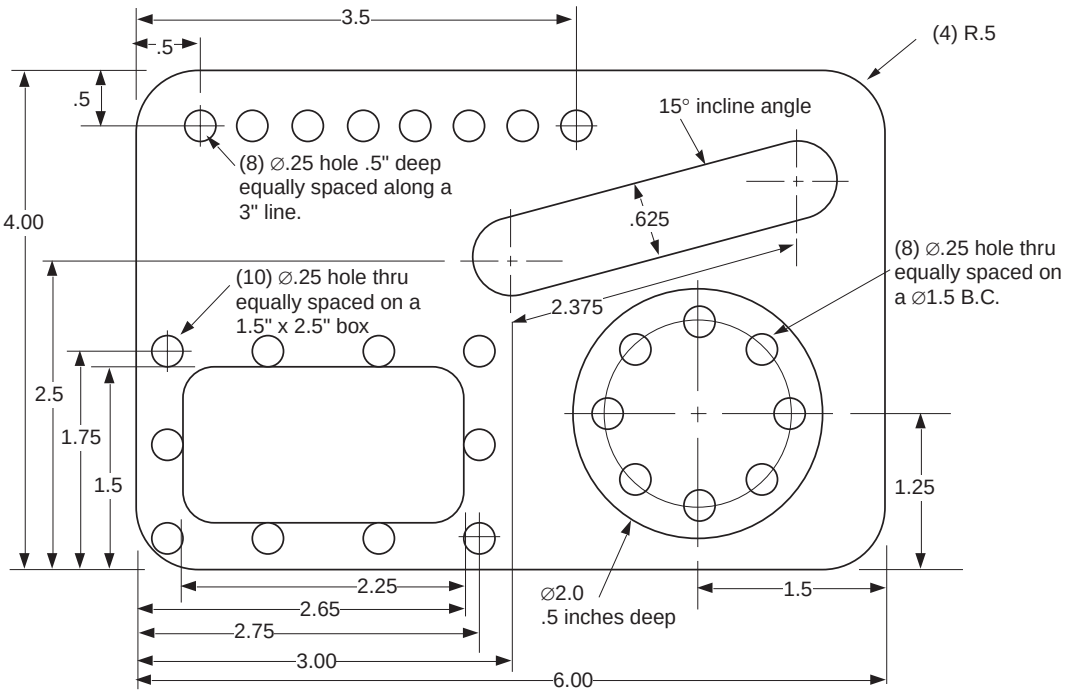


Figure 6-1 The blueprint for the part you are going to cut.

6.1 Cutting a Part on the EZTRAK

This tutorial describes cutting a simple part on the EZTRAK® using the **DO EVENT** mode to execute instructions one at a time. Each programmed event is given in this tutorial with a brief explanation. This tutorial assumes that you have already read the descriptions in chapters 3, 4, and 5 and are familiar with the basic operation of the EZTRAK®. This tutorial also assumes that you have some experience operating a manual turret mill, or you are at least familiar with the operation of one.

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

The part origin (X0, Y0) is placed at the center of the workpiece and Z0 is the top of the part.

It is highly recommended that you read through this tutorial completely before attempting to cut this part on the EZTRAK®.

I. Beginning the part

1) Turn on the EZTRAK®. Do this by turning the MAIN DISCONNECT on the cabinet at the back of the machine to the **ON** position.

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

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

4) The **MOV XYZ** screen is displayed automatically after the axis homing routine is executed (because the MOV XYZ key was pressed to initiate the homing sequence). Press the ESC key.

5) Press the **7 SET XYZ** key and enter X = **15.0** Y = **6.0** Z = **0.0**

6) Select the **3 MOV XYZ** key. Enter **0.0000** for X, **0.0000** for Y, and **0.0000** for Z and then press **Enter**. This moves the axes to the center of the worktable.

7) Put 1/2" End Mill in quill, secure workpiece in vise. You will need a block of material at least 6" X 4" X .75".

8) Use **JOG** mode to locate reference points. X0, Y0 are in the center of the part and Z0 is the top of the part.

9) Touch off top of part with End Mill and press **2 TLO = Z**. Enter **1** at **TOOL #** prompt and **.5** at **TOOL DIAMETER** prompt, raise quill above part.

II. Cutting the Outside Rectangle

1) Turn on spindle.

2) Select **2 DO EVENT** from the BASIC OPERATIONS screen, then press **5 MILL RECTANGLE**.

3) The starting point is at the right, front edge of the part. Complete the **MILL RECTANGLE** menu as follows:

	[0 = OUT, 1 = IN]	0	for outer rectangle
	T DIAM	.5	for tool diameter
	X center	0.	X center of rectangle
	Y center	0.	Y center of rectangle
*	Z clearance	.1	Z clearance point
	Z depth	.6	Z depth from clearance point
*	Z peck	0 or .6	can also be used for no peck
	X length	6	length of rectangle
	Y width	4	width of rectangle
	R blend	.5	fillet radius, 4 places
	approach	.5	ramp on, ramp off move
	allowance	.01	finish allowance
	F rough	10.	roughing feedrate
	F finish	20.	finishing feedrate

* Does not apply to 2 axis machines.

NOTE: Trailing decimal points can be omitted.

4) After inputting the last piece of data and pressing the **ENTER** key, press the + key to execute the move. The screen will show:

>>WAIT [CYCLE]
0010 RECT|CNTR OUT X0. Y0. Z.1 Z.6 Z0. X6. Y4. R.5 P.5 P.01 D.5 F10. F20.

5) The 3-axis EZTRAK will go to the Z clearance point, perform the cycle, and rapid back to the Z clearance point at the end.

The 2 axis EZTRAK will wait for the operator to check proper clearance, go to the calculated start point, wait for the operator to set proper depth, perform the cycle, wait for operator to clear the part, and then rapid back to a calculated start point.

III. Cutting the Rectangular Pocket

1) Select **5 MILL RECT** again and press **F4 RECT PKT**. Fill out the menu as follows:

PKT stepover	.25
T DIAM	.5
X center	-1.5
Y center	-1.
Z clearance	.1
Z depth	.6
Z peck	0 (or .6)
X length	2.0
Y width	1.0
R blend	.25
approach	.25
allowance	.01
F rough	10.
F finish	20.

2) After inputting the last piece of data and pressing the **ENTER** key, press the + key to execute the move. The screen will show:

>>WAIT [CYCLE]

0020 RECT|CNTR PKT X-1.35 Y-1. Z.1 Z.6 Z0. X2.0 Y1.0 R.25 P.25 P.01 P

3) The 3-axis EZTRAK® will go to the Z clearance point, perform the cycle, and rapid back to the Z clearance point at the end.

   The 2 axis EZTRAK will wait for the operator to check proper clearance, go to the calculated start point, wait for the operator to set proper depth, perform the cycle, wait for operator to clear the part, and then rapid back to a calculated start point.

IV. Cutting the Circular Pocket

1) Press **6 MILL CIRCLE** then press **F4 CIRCULAR POCKET**. Fill in the menu as follows:

PKT stepover	.2	
T DIAM	.5	
X arc cntr	1.5	
Y arc cntr	-.75	
Z clearance	.1	(Does not apply to 2-axis EZTRAK)
Z depth	.6	
Z peck	0 (or .6)	(Does not apply to 2 axis EZTRAK)
R radius	1	
approach	.25	
allowance	.01	
F rough	10.	
F finish	20.	

2) After inputting the last piece of data and pressing the **ENTER** key, press the + key to execute the move. The screen will show:

>>WAIT [CYCLE]
0030 CIRCLE PKT X1.5 Y-.75 Z.1 Z.6 Z0. R1. P.25 P.01 P.2 D.5 F10. F20.

3) The 3-axis EZTRAK® will go to the Z clearance point, perform the cycle, and rapid back to the Z clearance point at the end.

   The 2 axis EZTRAK will wait for the operator to check proper clearance, go to the calculated start point, wait for the operator to set proper depth, perform the cycle, wait for operator to clear the part, and then rapid back to a calculated start point.

V. Cutting the Slot

1) Next select **. MILL SLOT** and complete the menu as follows:

	T DIAM	.5	
	X ref arc cntr	0.	
	Y ref arc cntr	.5	
*	Z clearance	.1	(Does not apply to 2 axis EZTRAK)
	Z depth	.6	
*	Z peck	0 (or .6)	(Does not apply to 2 axis EZTRAK)
	lg = out to out	3.	length of slot
	slot width	.625	width of slot
	rotated angle	15.	
	F FEED	10.	

* Does not apply to 2 axis machines.

2) After inputting the last piece of data and pressing the **ENTER** key, press the + key to execute the move. The screen will show:

>>WAIT [CYCLE]
0040 SLOT X0. Y.5 Z.1 Z.6 Z0. P3. P.625 P15. D.5 F10.

3) The 3-axis EZTRAK® will go to the Z clearance point, perform the cycle, and rapid back to the Z clearance point at the end.

The 2 axis EZTRAK will wait for the operator to check proper clearance, go to the calculated start point, wait for the operator to set proper depth, perform the cycle, wait for operator to clear the part, and then rapid back to a calculated start point.

VI. Drilling Cycles

1) At this point you must change the tool to a 1/4" drill and enter a new TLO as you did at the beginning of this tutorial.

2) Turn on the spindle, select **2 DO EVENT**, then **9 BOLT CIRCLE** and enter the following:

	R bc radius	.75	
	X bc cntr	1.5	
	Y bc cntr	-.75	
*	Z clearance	-.4	(Does not apply to 2 axis EZTRAK)
	Z dr depth	.6	
*	Z peck	.3	(Does not apply to 2 axis EZTRAK)
	Z peck [2nd,...]	.2	
	A start	0.	
	# holes	8	
*	feedrate	10.	

* Does not apply to 2 axis machines.

3) After inputting the last piece of data and pressing the **ENTER** key, press the + key to execute the move. The screen will show:

```
>>WAIT [CYCLE]
0050 DR|BC R.75 XC1.5 YC-.75 Z-.4 Z.6 Z.3 X.2 A0. P8. F10.
```

4) The 3-axis EZTRAK will go to the Z clearance point, perform the cycle, and rapid back to the Z clearance point at the end. Hit **/ QUILL UP** at the end of the cycle to move the tool to a safe clearance point.

The 2 axis EZTRAK will wait for the operator to check proper clearance, go to the calculated start point, wait for the operator to set proper depth, allow the operator to perform the cycle, wait for operator to check proper clearance, and then rapid back to the calculated start point.

5) Select **8 DR BOX** and enter the following:

	X abs	-2.75	
	Y abs	-1.75	
*	Z clearance	.1	(Does not apply to 2 axis EZTRAK)
	Z dr depth	1.1	
*	Z peck	.3	(Does not apply to 2 axis EZTRAK)
*	Z peck [2nd,...]	.2	(Does not apply to 2 axis EZTRAK)
	X inc distance	2.5	
	Y inc distance	1.5	
	# X holes	4	
	# Y holes	3	
*	feedrate	10.	(Does not apply to 2 axis EZTRAK)

* Does not apply to 2 axis machines.

6) After inputting the last piece of data and pressing the **ENTER** key, press the + key to execute the move. The screen will show:

```
>>WAIT [CYCLE]
0060 DR|RECT  X-2.75 Y-1.75 Z.1 Z1.1 Z.3 Z.2 X2.5 Y1.5 P4. P3. F10.
```

7) The 3-axis EZTRAK® will go to the Z clearance point, perform the cycle, and rapid back to the Z clearance point at the end.

The 2 axis EZTRAK will wait for the operator to check proper clearance, go to the calculated start point, wait for the operator to set proper depth, allow the operator to perform the cycle, wait for operator to check proper clearance, and then rapid back to the calculated start point.

8) Select **7 DRILL ROW** and enter the following:

X ABS	-2.5	
Y ABS	1.5	
Z clearance	.1	(Does not apply to 2 axis EZTRAK)
Z dr depth	1.1	
Z peck	.3	(Does not apply to 2 axis EZTRAK)
Z peck [2nd,...]	.2	(Does not apply to 2 axis EZTRAK)

X inc distance	3.	
Y inc distance	0.	
# holes	8	
feedrate	10.	(Does not apply to 2 axis EZTRAK)

9) After inputting the last piece of data and pressing the **ENTER** key, press the + key to execute the move. The screen will show:

>>WAIT [CYCLE]
0070 DR|ROW ABS X-2.5 Y1.5 Z.1 Z1.1 Z.3 Z.2 X3. Y0. P8. P10.

10) The 3-axis EZTRAK® will go to the Z clearance point, perform the cycle, and rapid back to the Z clearance point at the end.

The 2 axis EZTRAK will wait for the operator to check proper clearance, go to the calculated start point, wait for the operator to set proper depth, allow the operator to perform the cycle, wait for operator to check proper clearance, and then rapid back to the calculated start point.

The part is now complete. Press **0** (**EXIT**) to leave the **DO EVENT** mode.

This is a good example of using 7 canned cycles in **DO EVENT** one at a time.
Production work would require a program, which would be nothing more than the same canned cycles above written in the **MDI** mode and executed in the **RUN** mode.

Chapter 7
MDI PROGRAMMING MODE

7.1 MDI PROGRAMMING

When the MDI (Manual Data Input) mode is selected, the screen displays:

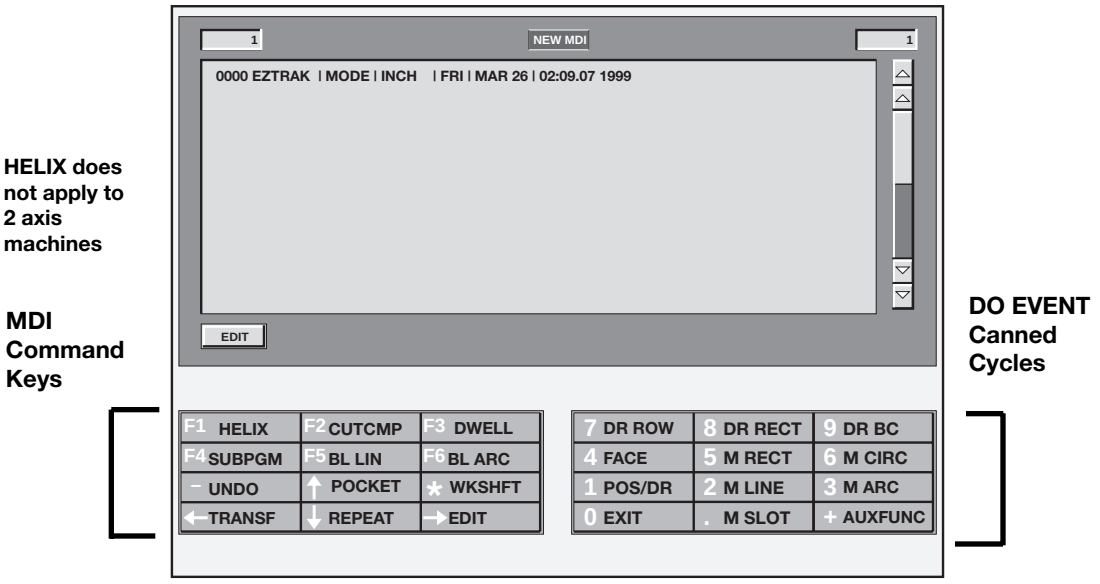


Figure 7-1

The MDI programming mode is used to program the EZTRAK to execute a series of machining operations. Each operation is programmed one at a time by selecting the type of operation, then entering the data necessary to execute the command. An **UNDO** command is available in case a line is programmed incorrectly. All of the cycles which are available in the DO EVENT mode can be programmed as single instructions in the MDI programming mode. In addition, cutter compensation, tangent arcs (blends), and graphic Geometry Help can be used to simplify the cutter path programming.

After a part is programmed it can be saved to the Disk On Module or a 3.5" diskette,

for later re-use.
When the MDI mode is entered, the first line of the part program is written out automatically. This line reads:

```
0000 EZTRAK 1 MODE|INCH  | MON JAN 30 14:21:00 1995
```

This determines that the program was written on the EZTRAK® and also establishes the unit of measure for the part program, as well as the date and time it was written. The unit of measure is determined **before** the MDI mode is selected. To program in millimeters, choose F2 MM in the Basic Screen before selecting MDI mode..

NOTE:
The first instruction in an EZTRAK program should be in absolute coordinates. The first instruction after a REPEAT command should also be programmed in absolute coordinates.

7.2 CONNECTIVE EVENTS

The EZTRAK® constantly monitors the tool position. Each operation in a part program must give a target location, a position which the tool is moving to, and the type of movement, such as a clockwise arc move, a linear move etc.

To simplify programming, the EZTRAK® always assumes that the tool is moving FROM its current position TO the target position, the location given in the programmed operation. It is important to be aware of the ending point in each of the canned cycles.

7.3 MDI COMMAND KEYS

HELIX does not apply to 2 axis machines →

F1 HELIX	F2 CUTCMP	F3 DWELL
F4 SUBPGM	F5 BL LIN	F6 BL ARC
- UNDO	⇧ POCKET	* WKSHFT
⇦ TRANSF	⇩ REPEAT	⇨ EDIT

The following pages describe the MDI command keys and their function.

HELIX does not apply to 2 axis machines

F1 HELIX

The **HELIX** command allows the operator to create and cut a helical pattern in 3-axis mode only. This command will cut a three-dimensional curve on a cylinder or cone at a constant angle.

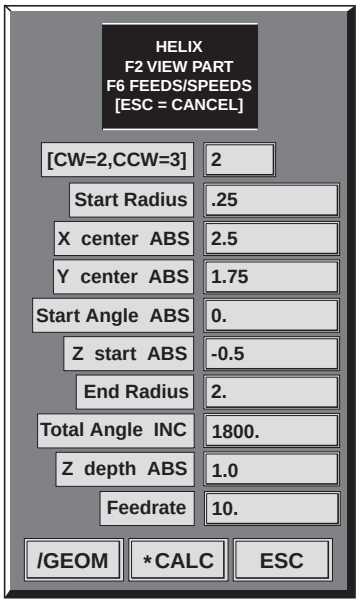


Figure 7-19

[CW=2,CCW=3]	the direction of the cut
Start Radius	the radius of the starting curve
X/Y cntr ABS	the X and Y center point of the starting curve
Start Angle ABS	the angle at which the cut will begin (0° to 360°)
Z Start ABS	the Z coordinate starting point
End Radius	the radius of the ending curve
Total Angle INC	the total incremental angular travel of the tool (1° – 65535°)
Z depth ABS	the endpoint of the helix in the Z axis (ABS)
feedrate	the rate of speed the ` tool cuts the part

The helix can be created in either a cylindrical format (Figure 7-20) or a conical format (Figure 7-21), sometimes called a spiral.

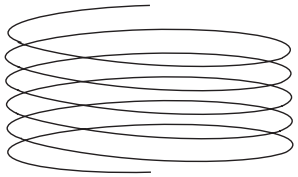


Figure 7-20

NOTE:
This command cannot be used with cutter compensation turned on. Also, a HELIX command cannot be TRANSFORMed in any way (mirrored, offset or rotated).

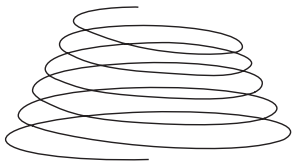


Figure 7-21

F2 CUTCOMP (CUTTER COMPENSATION)

In order to program the correct part shape, an allowance must be made for the cutter size. The tool path is offset by the cutter's radius so that the tool's cutting edge is placed on the programmed path. The calculations necessary for this allowance can be very difficult in some situations. The cutter compensation routines in EZTRAK greatly simplify this process.

Cutter compensation can be used in the following ways:

1. In EZTRAK the part shape can be programmed directly by the programmer. Using Cutter Comp, EZTRAK will automatically generate the tool path offsets.

2. If the tool wears or breaks and the same size tool is not available, Cutter Comp can be used to modify the program to allow for the changes in tool size.

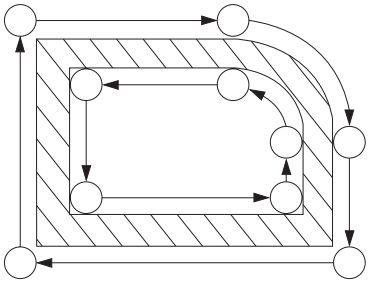
3. Up to two different comp values can be used to create roughing, semi-finish and finish passes.

The tool placement and cutting direction must be specified when using cutter compensation. The tool is placed to one side of the programmed path to allow for the tool's size. This placement must be on the correct side (LEFT or RIGHT) for the direction of the tool's movement. These placements are specified with TOOL LEFT, or TOOL RIGHT.

To determine which placement (TOOL LEFT or TOOL RIGHT) is correct, imagine that you are standing on the programmed path behind the tool, and that you are looking at the tool as it moves away from you. If the tool is on your left, then the TOOL LEFT placement is correct. If the tool is on your right, then the TOOL RIGHT placement is correct.

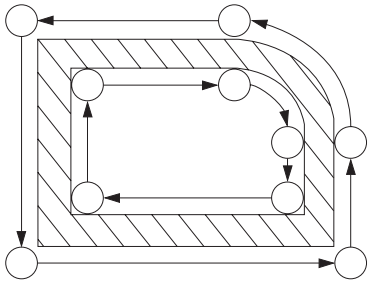
REMEMBER:

Tool placement with respect to the part is always determined by looking from behind the tool in the direction of motion.



TOOL LEFT

Figure 7-2



TOOL RIGHT

Figure 7-3

With standard right hand milling cutters:

TOOL LEFT = Climb Milling
TOOL RIGHT = Conventional Milling

RULES FOR USING CUTTER COMPENSATION

1. If the part program calls for cutting a concave or notch-like feature in the part, the cutter diameter must be no greater than the diameter or width of the feature to be cut. If the cutter diameter is larger than the width of the feature to be cut,

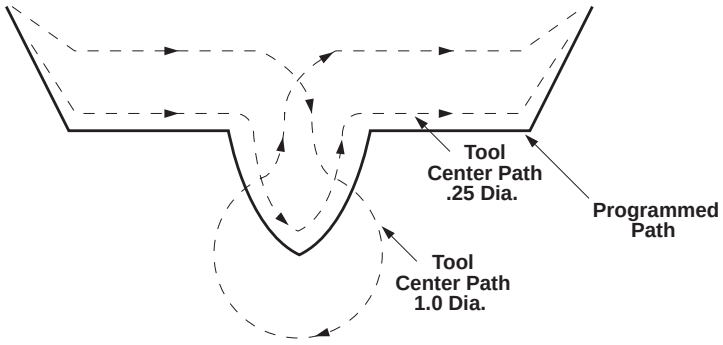


Figure 7-4

gouging occurs, as in Figure 7-4, below.

2. If the program calls for making a step less than the cutter radius, gouging occurs if corner rounding is turned on. Corner rounding may be turned off in the cutter compensation mode, to correct this problem. See Figure 7-5.

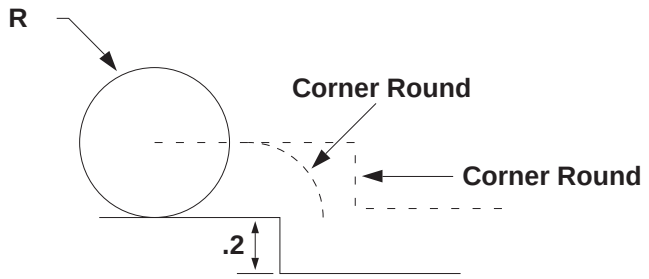


Figure 7-5

3. **Co-linear moves cannot be programmed**. Do not program two consecutive linear moves on the same line in cutter compensation.

4. **A circular Z move must not be programmed on the next line after turning Cutter Compensation on.**

```
0010 COMP|ON LFT D.5 X0 Y0 Z.1 Z-.1 P.5 F50
0020 BLEND|LN ABS X0 Y6 Z-.1 R1. CW F50
0030 LINE ABS X5.5 Y5.5 Z.1 F50
0040 BLEND|LN ABS X6. Y0 Z-.1 R1.CW F50
0050 LINE ABS X0 Y0 Z.1 F50
0060 COMP|OFF Z.5
```

Changes in Z depth are programmed in sequence numbers 30, 40 and 50. Operator prompt messages for Z moves will occur at positions A, B, and C in Figure 7-6.

NOTE: It is **not** valid to program a Z move in an **ARC** or **BLEND ARC** instruction on the next line after turning on Cutter Compensation.

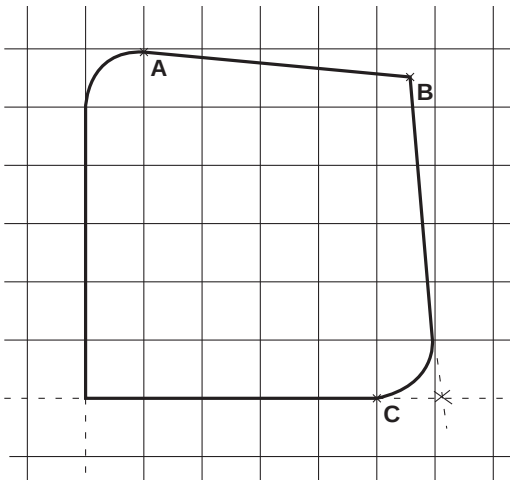


Figure 7-6

5. **Only M LINE, M ARC, BL LIN, and BL ARC moves are allowed**. When cutter compensation is turned on the command keys are changed so that only the legal instructions for the cutter comp mode are displayed. **Do not add any other program lines between COMP ON and COMP OFF**. To add illegal cutter comp lines later will cause errors in execution, and will halt the program prematurely.

STARTING UP CUTTER COMPENSATION

Starting the cutter compensation requires some special tool movements. These movements are calculated and created by EZTRAK; however, it is important to understand how the moves are created so that the part can be designed and cut correctly.

The tool moves to a point offset from the path by the tool radius plus the allowance. The tool approaches the path with an arc move that is tangent to the path at the start point. The radius of the arc is half of the allowance value. This is shown in Figure 7-7 and 7-8 below, for a linear move (M LINE, or BL LIN), and an arc move (M ARC or BL ARC).

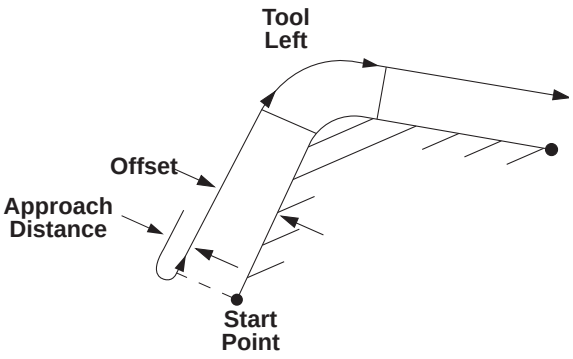


Figure 7-7

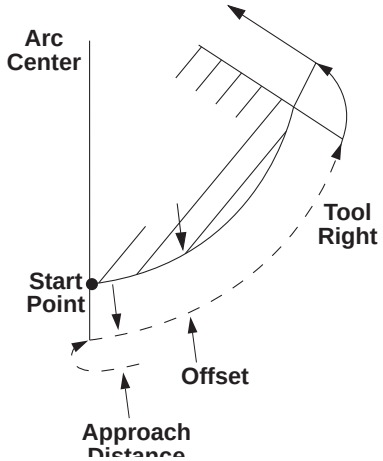


Figure 7-8

F2 COMPON

This instruction turns on the cutter compensation feature creating tool movements as described above.

LFT/RGT	the direction of cutter offset
1st DIA.	the tool diameter for the first cut
approach	the approach distance
X_Y_	the start point of the first line or arc to be compensated
Z approach	the Z clearance for positioning.
Z mill ABS	the milling depth.
Feedrate	the feed rate to the start point.

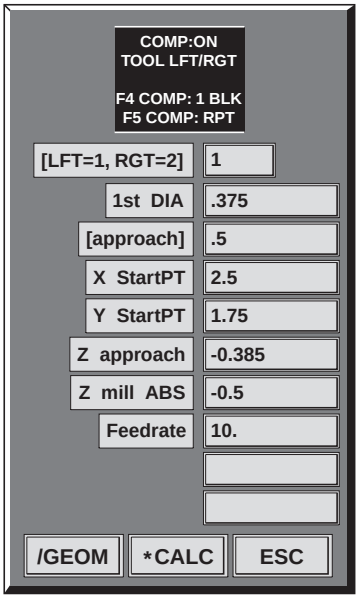


Figure 7-9

This command has two optional modes, which are selected by pressing the

F4 COMP: 1 BLK and
F5 COMP:RPT keys.

The **F4** key selects the **COMP 1BLK** command which turns on the cutter compensation feature for only one programmed instruction. The ramp on and ramp off movements are created the same as the usual COMP ON command. Do **not** put a **COMP OFF** line after COMP 1BLK. The program for Figure 7-10 would look like this:

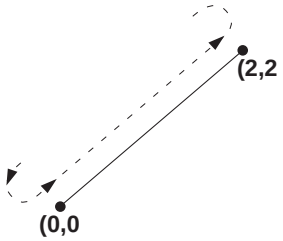


Figure 7-10

```
0010 COMP|1BLK LFT D.5 X0 Y0 Z-.1
     Z-.25 P.375 F10.
0020 LINE ABS X2.0 Y2.0 Z-.25 F10.
```

The **F5 COMP:RPT** mode can create both roughing and finishing passes with two different compensated diameters and feedrates. The instructions between **COMP ON** and **COMP OFF** are executed once, then the tool rapids back to the start point before the finish

pass, using the second diameter and feedrate.

2nd DIA is the comp diameter of the second cut
2nd FEED is the feedrate of the second cut

CORNER ROUNDING IN CUTTER COMPENSATION

The angle of intersection created by two blocks of motion commands as measured on the workpiece side create an “inside” or “outside” corner. An inside-corner occurs when the angle is over 180 degrees, an outside-corner occurs when the angle is less than 180 degrees. (The two moves may also be tangent at the point of intersection).

If an outside corner occurs, a rounding arc is automatically blended through the point of intersection tangent to the two programmed paths. Figures 7-11 and 7-12 show inside corners which have an intersection angle greater than 180°. Figures 7-13 and 7-14 show outside corners which have an intersection angle less than 180°.

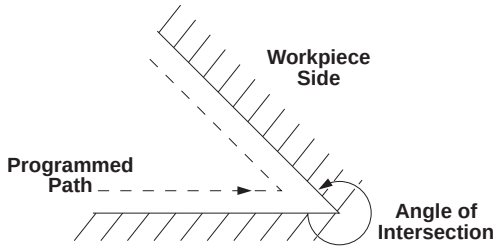


Figure 7-11

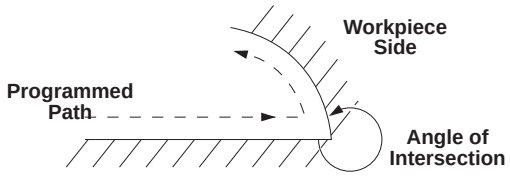


Figure 7-12

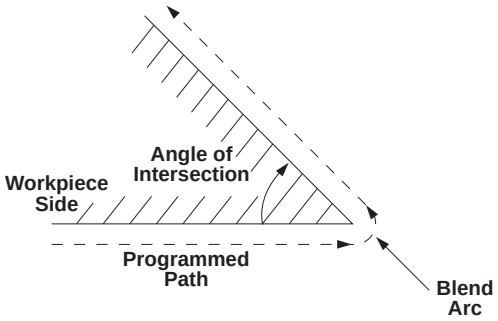


Figure 7-13

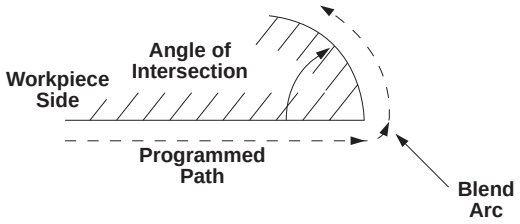


Figure 7-14

TURNING CORNER ROUNDING ON/OFF

When programming in the Cutter Comp mode, the corner rounding feature can cause gouging when cutting steps or notches smaller than the cutter radius. To avoid this problem, the corner rounding feature should be turned off while cutter comp is on.

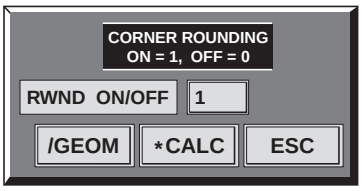


Figure 7-15

To turn corner rounding off, select the right arrow key **C RND** after COMP ON has been programmed.

EXITING CUTTER COMPENSATION

EZTRAK also uses a ramp off move to exit from the cutter compensation mode. The following shows examples of the cutter compensation exit moves.

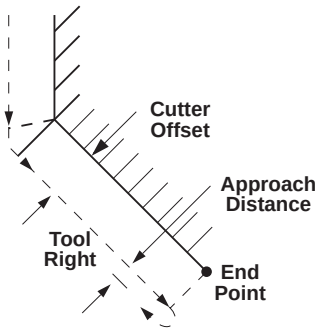


Figure 7-16

If the last compensated move is a line, the compensation exit moves appear similar to Figure 7-16**.** The tool moves to the end point of the programmed path, and then exits by making an arc move away from the path. The arc move has a radius value equal to half the allowance value.

If the last compensated move is an arc, the cutter comp exit moves appear similar to Figure 7-17. The tool moves to the end point of the programmed path, and then exits by making an arc move away from the path. The arc move has a radius value equal to half the allowance value.

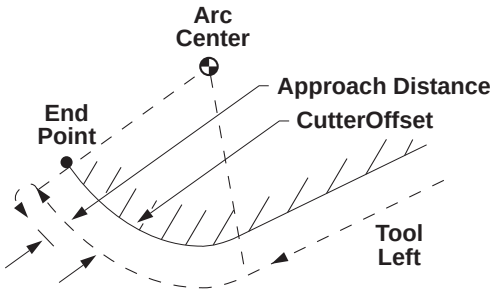


Figure 7-17

F2 CMPOFF

The **CMPOFF** command turns cutter compensation off. This move allows the user to enter a Z value so that the tool can be withdrawn from the part at the end of the cutter compensation.

NOTE:
Do not use COMP OFF after a COMP 1 BLOCK command.

Figure 7-18

F3 DWELL

The **DWELL** command allows the operator to create a pause in the program that will keep a tool in position for the desired number of seconds.

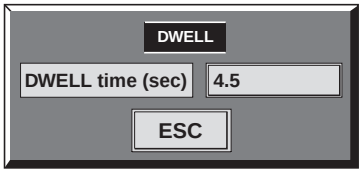


Figure 7-22

DWELL time (sec) is the number of seconds of dwell

F4 SUBPGM

The **SUBPGM** command allows one program to call and execute another program as long as both are accessible in memory. There are no limits to the number of programs that can call each other. The **SUBPGM** command may be used with the **TRANSF** (**ROTATE**, **OFFSET**, or **MIRROR**) commands. An example of this is given in the section which describes the **TRANSF** command.

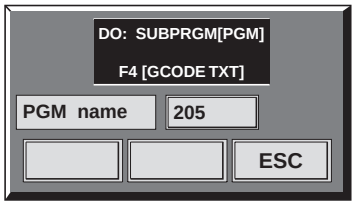


Figure 7-23

The **SUBPGM** command requires only the name of the program that is being called. Either a **.PGM** or a **.TXT** file may be used as a sub-program. Press the **F4** key to enter the name of **.TXT** file.

BLENDS (F5 BL LIN, F6 BL ARC)

EZTRAK provides commands in the MDI programming mode for placing tangent arc moves between lines and arcs (or any combination of lines and arcs). These moves are called **BLENDS** and are only available in the MDI programming mode. The two commands which program a blend are **BL LIN** and **BL ARC**.

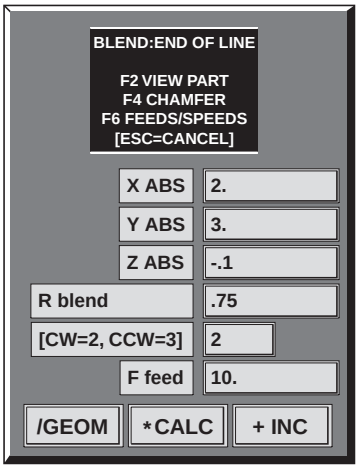


Figure 7-24

To program a blend, the intersection of the two connecting entities must be given in the **X_Y_** fields. This point can be calculated using the * **CALC** or the **/GEO** utilities.

NOTE: The **BL LIN** and **BL ARC** commands cannot be programmed after a **COMP1 BLK** command, because these commands require two instructions to complete.

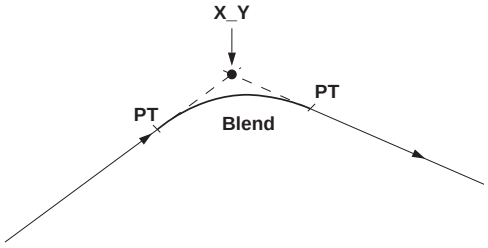


Figure 7-25

F5 BL LIN

This command places a blend arc after the programmed line. The **BL LIN** command requires the following parameters.

X_Y ABS	the intersection of the LINE and the next LINE or ARC in the tool path (see Figure 7-25).
Z ABS	Z depth (signed).
R radius	the blend radius.
CW/CCW	the direction of the blend arc
F feed	the feedrate.

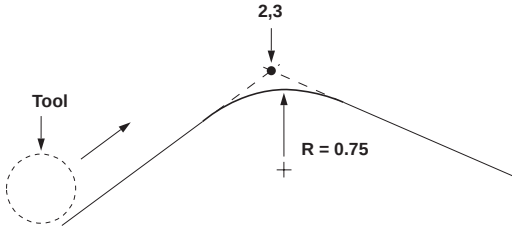


Figure 7-26

For example, this instruction:

```
0020 BLEND|LN ABS X2. Y3. R.75 CW F10.
```

cuts the path shown in Figure 7-26.

CHAMFER

The **BL LIN** command can also program a **CHAMFER**, a straight line that connects two linear moves. The **CHAMFER** is programmed by selecting the **BL LIN** command, then pressing the **F4** key. The **CHAMFER** command requires the following parameters.

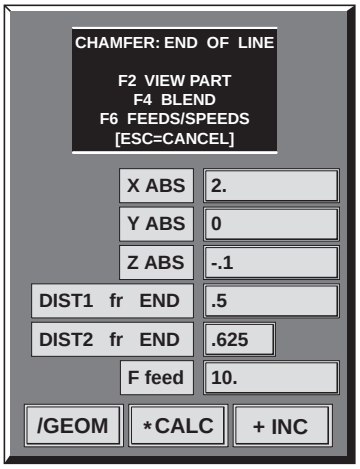


Figure 7-27

X ABS Y ABS	the **intersection** of the LINE and the next move in the tool path (see Figure 7-28).
Z ABS	the depth of the cut (signed).
DIST1 fr END	the distance from PT1 to the intersection as shown in Figure 7-28.
DIST2 fr END	the distance from the intersection to PT2 as shown in Figure 7-28.
F feed	the feedrate.

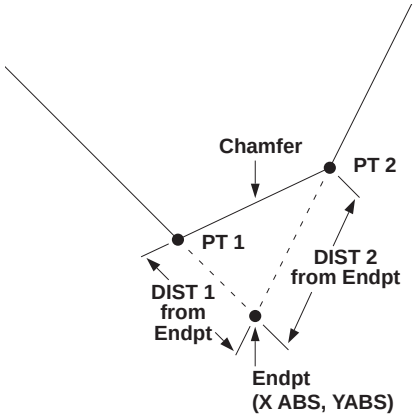


Figure 7-28

The instructions to program the example shown in Figure 7-29, would be:

```
10 CHAMFER ABS X2. Y0. Z-.1 P.5  P.625 F10.
20 LINE ABS X3 Y2 Z-.1 F10.
```

The first line programs the endpoint, where the two lines would intersect. It also sets the distances from this point, where the chamfer connects the two lines. The second line programs the linear move to the point (3,2).

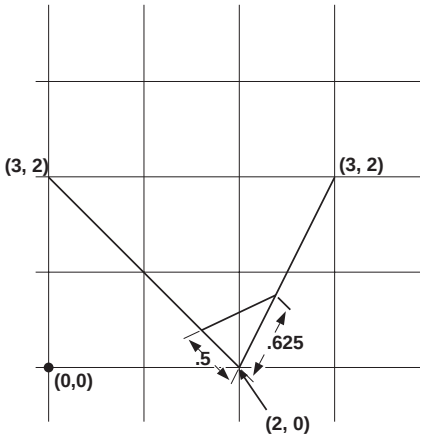


Figure 7-29

F6 BL ARC

This command places a blend arc at the end of the programmed arc. The **BL ARC** command requires the following parameters.

CW/CCW	the direction of the programmed ARC
X ABS Y ABS	the intersection point of the ARC and the next LINE or ARC in the path (see Fig 7-31.)
Z ABS	the Z depth value (signed)
XC_YC_	the ARC center point
or **R arc radius**	the ARC radius
R blend	the blend radius
CW/CCW_	the direction of the blend arc
F feed	the feedrate

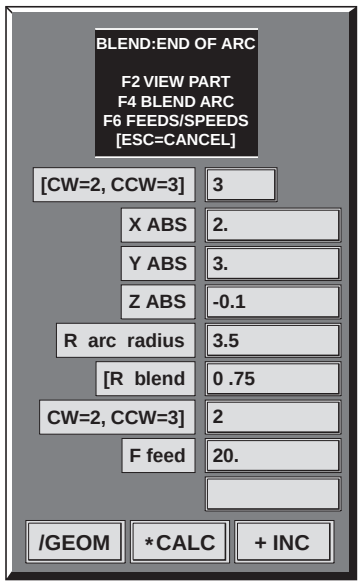


Figure 7-30

In this example, the tool cuts the arc to **PT1**, then cuts the blend arc, and ends at **PT2**.

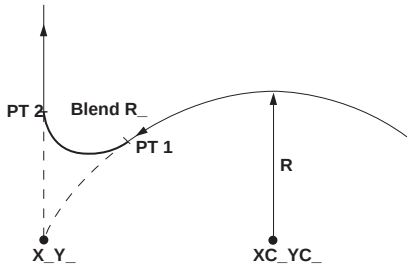


Figure 7-31

The instruction to cut the path in Figure 7-32 would be:

```
0040 BLEND|ARC|RADIUS ABS CCW X2. Y3.
R3.5 R.75 CW F20
```

NOTE

You cannot end a contour with a blend line or blend arc. A mill line or mill arc needs to follow the last blend command.

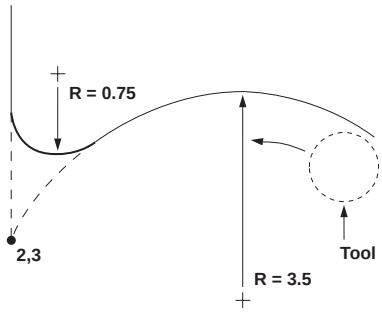


Figure 7-32

– UNDO

If any line is programmed incorrectly, it can be deleted by pressing the -UNDO key. This automatically deletes the last programmed line. This key can be used repeatedly.

↑POCKET

To create an irregularly shaped pocket, you first need to define the *path* or shape of the pocket.

1. In MDI, select **↑ POCKET**. This will call up the DEFINE PATH window (Figure 7-33).

2. In DEFINE PATH, enter a Path Number [ ], a clockwise (2) or counterclockwise (3) direction, and an X, Y starting point. This will enter the first path line in the program (see Line 30 in the program example.).

3. The selections for valid path operations will then appear: [2 M LINE], [3 M ARC], [F5 BL LIN], [F6 BL ARC], [F2 PTH OFF] and [-UNDO].

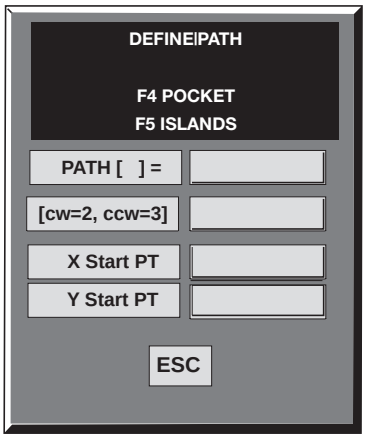


Figure 7-33

Rules:

1. A minimum of 3 lines, 2 arcs, 1 arc and 1 line or circle determine a path.
2. The final end point in the path definition must be the same as the starting point.
3. A path definition cannot end in a blend line or blend arc move.
4. A path definition cannot be defined inside another path definition
5. Colinear moves cannot be programmed. Do not program two consecutive lines with identical instructions in the same path.

For example, the following program will produce the pocket defined in Figure 7-34:

```
0010 || TOOLCHG T1
0020 || SPINDLE ON S1000
0030 PATH[1] CW X3.0 Y-2.0
0040 BLEND|LN ABS X0. Y-2.0 Z0. R.625 CW F10.
0050 BLEND|LN ABS X0. Y4.0 Z0. R.925 CW F10.
0060 BLEND|LN ABS X6.0 Y-2.0 Z0. R.35 CW F10.
0070 LINE ABS X3.0 Y-2.0 Z0. F10.
0080 END|PATH
```

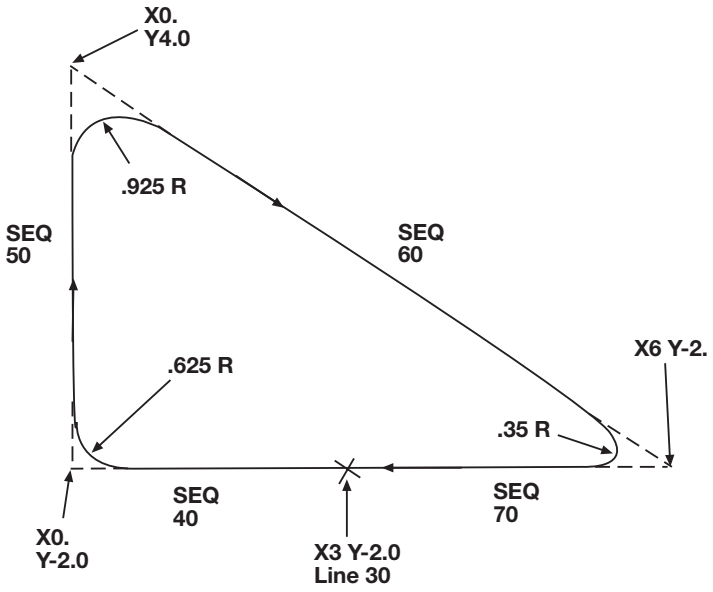


Figure 7-34

4. After the path definition, select [↑POCKET] again and then [F4 POCKET], which will bring up the Pocket Parameter window (Figure 7-35). Enter the desired values for the Pocket Parameters line.

Pocket Parameters

PATH [ ] = The path number.

(1=horiz, 2= vert) The zigzag direction. An entry of 1 (horizontal) means that the tool will mill out the interior of the part shape, moving back and forth in the X direction.An entry of 2 (vertical) means that the tool will mill out the interior of the part shape, moving back and forth in the Y direction.

T DIA The diameter in inches or mm that the tool moves before beginning a cutting pass.

Stepover The unsigned amount in inches/mm that the tool will move horizontally or vertically before making a cutting pass the length or width of the part.

Allowance Amount of stock in inches or mm the tool leaves along the inside perimeter during roughing passes, to be removed later during the finish pass.

Z Clearance The safe Z position that the cutter rapids to before moving in the X or Y direction, or at the start and end of each Z step.

Z ABS The signed final Z position of the pocket floor for the finish pass.

Z Step The distance to lower the cutting tool before starting each roughing pass.

F Rough The feedrate used for the roughing passes.

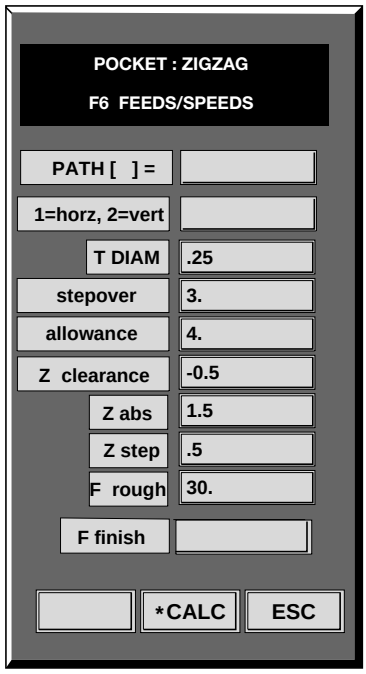


Figure 7-35

Note that the Pocket Parameter line must be specified *after* the path definition:

```
0000 EZTRAK    1 MODE|INCH  |TUE SEP 21 19:47:03 1999
0010 || TOOLCHG T1
0020 || SPINDLE ON S1000
0030 PATH[1] CW X3.0 Y-2.0
0040 BLEND|LN ABS X0. Y-2.0 Z0. R.625 CW F10.
0050 BLEND|LN ABS X0. Y4.0 Z0. R.925 CW F10.
0060 BLEND|LN ABS X6.0 Y-2.0 Z0. R.35 CW F10.
0070 LINE ABS X3.0 Y-2.0 Z0. F10.
0080 END|PATH
0090 POCKET|ZIGZAG|H PATH[1] D.375 P.2 P.01 Z.1 Z-.2 Z-.2 F25. F25.
0100 || END|PRGM@CLR
```

F5 ISLANDS

With this feature you can cut as many as 15 islands inside a pocket.

To create an island within a pocket, follow the procedure for creating a pocket described in the previous section. Then create another path that defines the island.

When the second path has been defined, select ↑POCKET again and choose F5 ISLANDS in the Define Path window (Figure 7-36). Figure 7-37 will appear. Enter the number of the Island Path. Note: A pocket and island cannot have the same path number.

NOTE: In the program, the path lines must come before the Pocket Parameters line., and the Pocket Islands line (Figure 7-37; Line 140 in the program example, next page) must come immediately *after* the Pocket Parameters line.

The desired path(s) must be defined by PATH START and END|PATH blocks prior to the POCKET and ISLAND blocks.

The programming example on the next page defines an island inside a pocket (Figure 7-38).

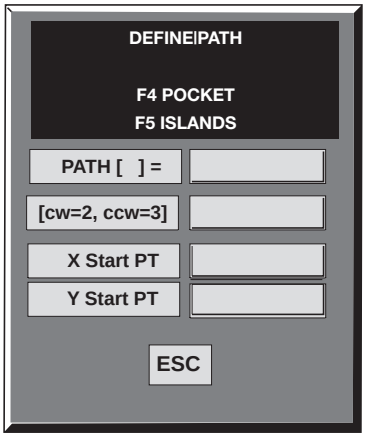


Figure 7-36

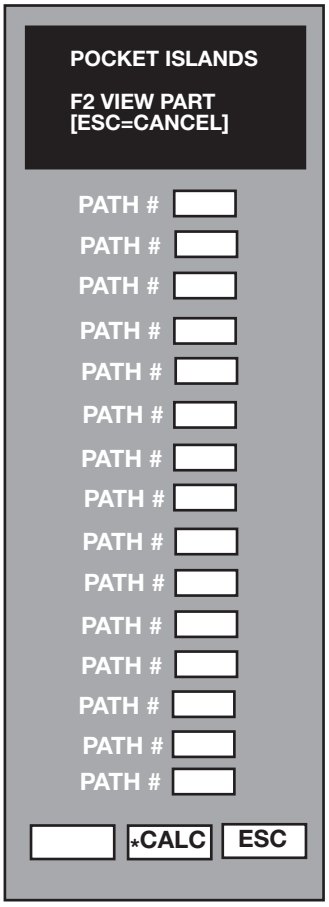


Figure 7-37

```
0000 EZTRAK    1 MODE|INCH  |SAT MAR 10 02:48:05 2001
0010 || TOOLCHG T1
0020 || SPINDLE ON S1000
0030 PATH[1] CW X3. Y-2.
0040 BLEND|LN ABS X0. Y-2. Z0. R.625 CW F10.
0050 BLEND|LN ABS X0. Y4. Z0. R.925 CW F10.
0060 BLEND|LN ABS X6. Y-2. Z0. R.35 CW F10.
0070 LINE ABS X3. Y-2. Z0. F10.
0070 END|PATH
0080 PATH[2] CW X2. Y-1.
0090 BLEND|LN ABS X1. Y-1. Z0. R.25 CW F10.
0090 BLEND|LN ABS X1. Y1.5 Z0. R.2 CW F10.
0100 BLEND|LN ABS X3. Y-1. Z0. R.2 CW F10.
0110 LINE ABS X2. Y-1. Z0. F10.
0120 END|PATH
0130 POCKET|ZIGZAG|H PATH[1] D.375 P.2 P.01 Z.1 Z-.2 Z-.2 F10. F10.
0140 ISLANDS P2 P0. P0. P0. P0 P0. P0. P0. P0. P0. P0. P0. P0. P0. P0.
0150 || END|PRGM@CLR
```

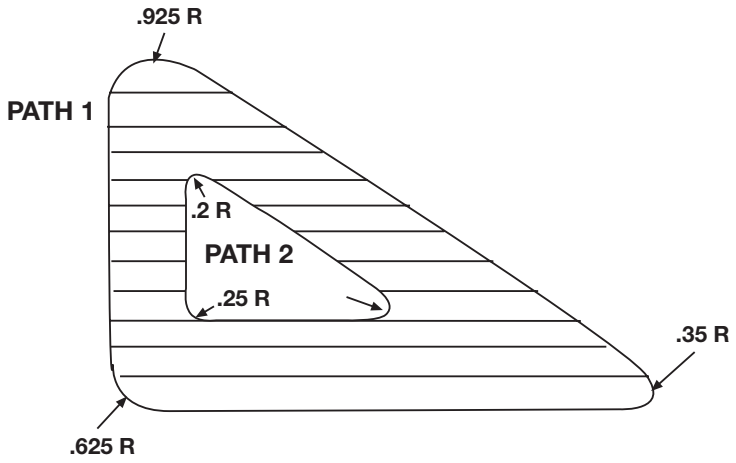


Figure 7-38

Select F2 VIEW PART to see a graphic preview of the programmed paths. The screen should display the island within the pocket, and the horizontal or vertical lines of the zigzag milling operation.

* WKSHFT (WORKSHIFTS)

EZTRAK has the capability to execute up to six workshifts, meaning that you can program the EZTRAK to machine as many as six parts during the same work cycle. To set or edit workshifts, see Chapter 4, JOG.

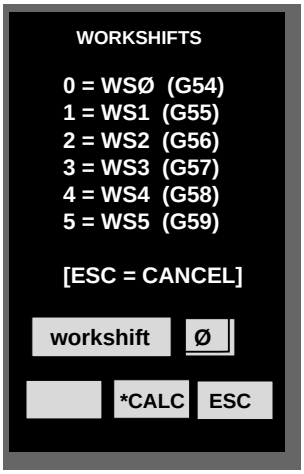


Figure 7-36

←TRANSF

This command can be used in four different ways.

1) It is used to move the work coordinate system to another location (see **OFFSET**).

2) It is used to scale the programmed part larger or smaller (see **SCALE**)**[F3]**

3).It is used to create a mirrored image of the part (see **MIRROR**).**[F4]**

4) It is used to rotate the work coordinate system (see **ROTATE**) **[F5].**

4)

These commands are described as follows.

OFFSET

To use the **TRANSF** command to move the work coordinate system to another location (**OFFSET**), enter the offset values for the X, Y, and Z directions.

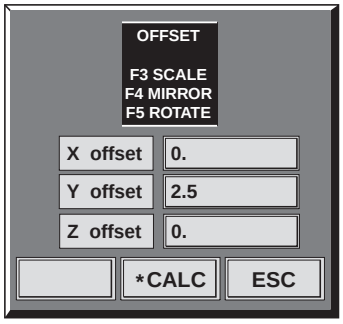


Figure 7-37

X Offset	the distance to be offset in the **X** axis
Y Offset	the distance to be offset in the **Y** axis
Z Offset	the distance to be offset in the **Z** axis

Once a translation (**OFFSET**) has been commanded, it remains in effect until it is reset. To reset the translation, use the **TRANSF** (**OFFSET**) command again, and enter **0** for the **X**, **Y** and **Z** offsets. This must be done before the end of the program.

```
0000 EZTRAK 1 MODE:INCH
0010 RAPID ABS X0.00 Y0.00 Z0.00
0020 LINE ABS X0.00 Y0.00 Z-1.00 F30.
0030 LINE ABS X1. Y0.00 Z-1.00 F30.
0040 ARC|CNTRPT ABS CCW X1. Y2. Z-1. XC1.
     YC1.0 F30.
0050 LINE ABS X0. Y0. Z-1. F30.
0060 TRANSLATE OFFSET X0 Y2.5 Z0.
0070 RAPID ABS X0.00 Y0.00 Z0.00
0080 LINE ABS X0.00 Y0.00 Z-1.00 F30.
0090 LINE ABS X1. Y0.00 Z-1.00 F30.
0100 ARC|CNTRPT ABS CCW X1. Y2. Z-1.0. XC1.
     YC1.0 F30.
0110 LINE ABS X0. Y0. Z-1.0. F30.
0120 TRANSLATE OFFSET X0 Y0 Z0
```

Lines 10-50 cut the part in the base coordinate system. Line 60 offsets the coordinate system 2.5 inches in the Y direction. Lines 70-110 cut the part again. Line 120 resets the work coordinate system back to the original location. If another part were to be cut at the location **7.0,-3.0** the **TRANSF** command would be used again with the offsets **7.0** and -**3.0**.

MIRROR

When the **F4** key is pressed in the **TRANSF** command, a mirror image of the part can be created. The axis of reflection is the current tool location when the **MIRROR** command is executed. (Note in the example below that the tool is positioned back to **2.0,1.0** at the end of program 1, before each **MIRROR** command.)

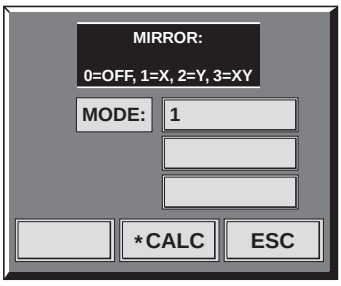


Figure 7-38

MIRROR MODE
Enter the desired reflection number.

1 reflects across the X axis,
2 reflects across the Y axis,
3 reflects across both axes.

Once a translation has been commanded, it remains in effect until it is reset. To reset the translation, use the **TRANSF** command again, and enter **0** for the **MIRROR MODE**. This must be done before the end of the program.

EXAMPLE OF MIRROR IMAGE (WITH SUBPROGRAM)

Program 1 defines the shape

```
0000 EZTRAK 1 MODE:INCH
0010 RAPID ABS X2. Y1. Z.1
0020 RAPID ABS X3. Y1.25 Z.1
0030 LINE INC X0. Y0. Z0 F10.
0040 LINE INC X0. Y.25 Z0 F10.
0050 LINE INC X.25 Y0. Z0 F10.
0060 LINE INC X-.25 Y0. Z0 F10.
0070 LINE INC X0. Y.25 Z0 F10.
0080 LINE INC X.3 Y0. Z0 F10.
0090 RAPID ABS X2. Y1. Z.1
```

Program 2 machines the various images

```
 0000 EZTRAK 1 MODE:INCH
 0010 RAPID ABS X-2. Y2. Z.1
 0020 DO|SUBPRGM [PGM] 1
 0030 TRANSLATE MIRROR X
 0040 DO|SUBPRGM [PGM] 1
 0050 TRANSLATE MIRROR XY
 0060 DO|SUBPRGM [PGM] 1
 0070 TRANSLATE MIRROR Y
0080 DO|SUBPRGM [PGM] 1
0090 TRANSLATE MIRROR OFF
0100 RAPID ABS X-2. Y2. Z.1
```

NOTE:
Program 2 must be the active program in order to run the subprogram. If a change is made to Program 1, Program 2 must be reloaded into the editor and saved in order to update changes made to Program 1.

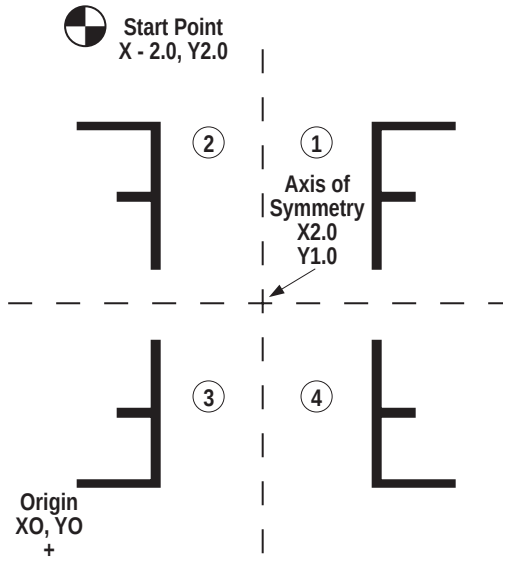


Figure 7-39

ROTATE

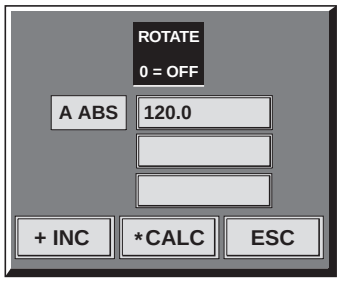


Figure 7-40

The **ROTATE** command is used to rotate the orientation of the part program through a specified angle. The center of the rotation is the part program origin (X = 0, Y = 0). After using the **ROTATE** command, the orientation of the part must be reset to its original position. This is done by calling the **ROTATE** command again, in the **absolute** mode, with an angle of zero. **ROTATE** is most often used with **REPEAT**.

A ABS This parameter gives the angle that determines how far the part orientation is rotated.

INC / ABS This changes the mode of measurement of the angle of rotation.

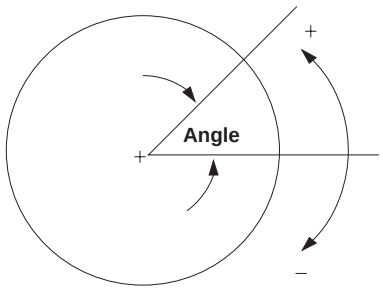


Figure 7-41

NOTE:
When the Rotate command is used in the absolute mode, the angle is measured from the positive X axis, as shown in Figure 7-41.

A simple example of using the **ROTATE** command is given below.

```
0000 EZTRAK 1 MODE:INCH
0010 RECT:CNTR PKT X0.00 Y3.00 Z0. Z-.5 Z-.5 X3.Y2.
     R.375 P.375 P0.5 P.125 D.25 F10.F15.
0020 ROTATE|INC A120.0
0030 RECT:CNTR PKT X0.00 Y3.00 Z0. Z-.5 Z-.5 X3. Y2.
     R.375 P.375 P0.5 P.125 D.25 F10. F15.
0040 ROTATE|INC A120.0
0050 RECT:CNTR PKT X0.00 Y3.00 Z0. Z-.5 Z-.5 X3. Y2.
     R.375 P.375 P0.5 P.125 D.25 F10. F15.
0060 ROTATE|ABS A0.
0070 AUXFUN T2 M2
```

This program cuts a box pocket at the location 0,3 (line 0010). The program then rotates the orientation of the part by 120° (line 0020) and a second pocket is cut (line 0030). Line 0040 causes another rotation, and line 0050 cuts a third pocket. The results of this program are shown in Figure 7-42. Line 0060 resets the orientation of the part to its original position.

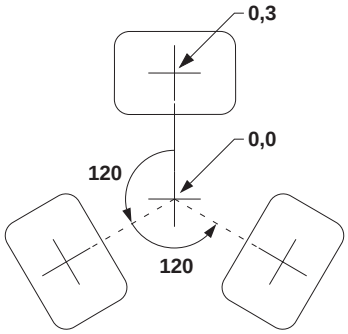


Figure 7-42

SCALE

Pressing the **F3** key in the **TRANSF** command allows the user to create a part that is a percentage larger or smaller than the programmed part.

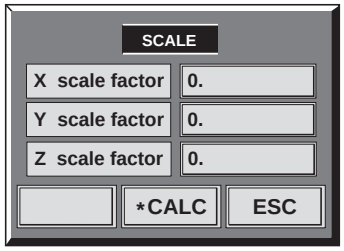


Figure 7-43

X Scale factor : the percentage of change in the X axis.

Y Scale factor : the percentage of change in the Y axis.

Z Scale factor: the percentage of change in the Z axis.

The total range of reduction and enlargement is from .001 to 99.999 times. If 1.0 is entered for a scale factor, then the part will be machined at the programmed size (100%). If a scale factor of less than 1.0 is entered, then the part will be machined at that percentage less than the programmed size.

For example: the X scale factor entered is .82, the Y scale factor is 1.0, and the Z scale factor is 1.0; the result is a part where all dimensions in the X axis are 82% of the programmed size.

If a scale factor of greater than 1.0 is entered, then the part will be machined at that percentage greater than the programmed size.

For example: the X scale factor entered is 3.0, the Y scale factor is 1.0, and the Z scale factor is 1.0; the result is a part where all dimensions in the X axis are 3 times (or 300%) the programmed size.

NOTE:
Cutter Compensation cannot be used with the SCALE command. If circle data is input, the X and Y scale factors must be the same.

↓ REPEAT

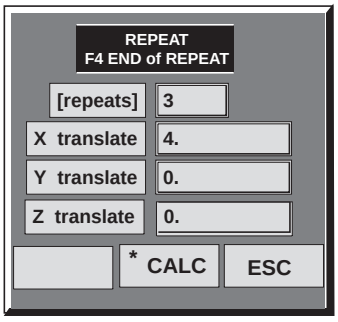


Figure 7-44

The **REPEAT** command is used to create a loop in a part program that will cause one or more instructions to be repeated a specified number of times. A shift can also be entered, so that the instructions are repeated at another location on the part. The **REPEAT** command is placed at the beginning of the set of instructions to be repeated (enter the number of times the loop is repeated) and the **END|REPEAT** command is placed after the last instruction to be repeated.

repeats the number of times the loop is executed.

X translate the shift in the X axis, between each execution of the loop.

Y translate the shift in the Y axis between each execution of the loop.

Z translate the shift in the Z axis between each execution of the loop

An example of the **REPEAT** command is given below.

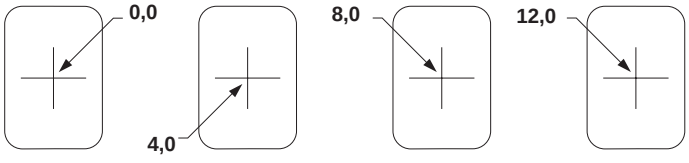


Figure 7-45

```
0000 EZTRAK 1 MODE:INCH
0010 REPEAT 3 X4. Y0. Z0.
0020 RECT:CNTR PKT X0.00 Y0.00 Z.1 Z.5 Z.25 X2. Y3.
 R.375 P.375 P0.5 P.125 D.25 F10. F15.
0030 END|REPEAT
0040 AUXFUN M2
```

This program contains a loop that has one instruction, to cut a box pocket shape (line 0020). The first instruction (line 0010) establishes a loop that will be executed four times, with a shift of 4 inches in the X direction, and zero inches in the Y direction between each execution of the loop. Since the loop is executed four times, this program cuts four box pockets, and looks like Figure 7-45.

NOTE:

The instruction is executed once, then repeated three times, resulting in four executions. The number of repeats should be the total number of executions minus one.

→EDIT

This key accesses the Part Program Editor, which displays a list of files available for editing. See Chapter 8, EDIT MODE.

7.4 CANNED CYCLES

The following canned cycles are available in MDI. See Chapter 5, DO EVENT for definitions.

7 DR ROW	8 DR RECT	9 DR BC
4 FACE	5 M RECT	6 M CIRC
1 POS/DR	2 M LINE	3 M ARC
0 EXIT	. M SLOT	+ AUXFUNC

Figure 7-46

Of these canned cycles, the following work differently in MDI:

.M SLOT
4 FACE
+ AUXFUNC
7 DR ROW

Their functions in MDI are described on the following pages.

. M SLOT (SLOT ARC)

Z Clearance and Z Peck do not apply to 2 axis machines.

In the MDI mode the **.M SLOT** command can be used to create an arc slot.

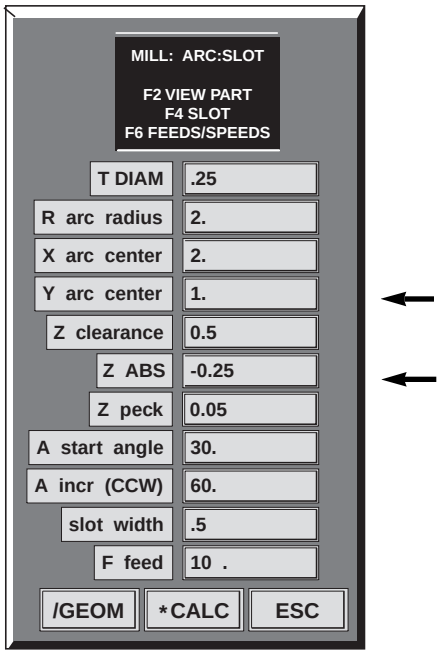


Figure 7-47

T DIAM	the diameter of the tool.
R arc radius	the radius of the slot arc.
X arc cntr	the X coordinate of the center point of the arc.
Y arc cntr	the Y coordinate of the center point of the arc.
Z clearance	the clearance height above the part.
Z ABS	the Z coordinate showing the milling depth.
Z peck	the peck depth.
A start angle	the start angle of the slot arc.
A incr (CCW)	the swath of the slot arc.
slot width	the edge to edge width of the slot.
F feed	the feedrate in inches per minute of the milling operation.

For example, the instruction:

```
SLOT|ARC R2. XC2. YC1. Z0.5 Z-0.25
      Z0.05 A30 A60 P.5 D.25 F10.
```

will cut the arc shown in Figure 7-48.

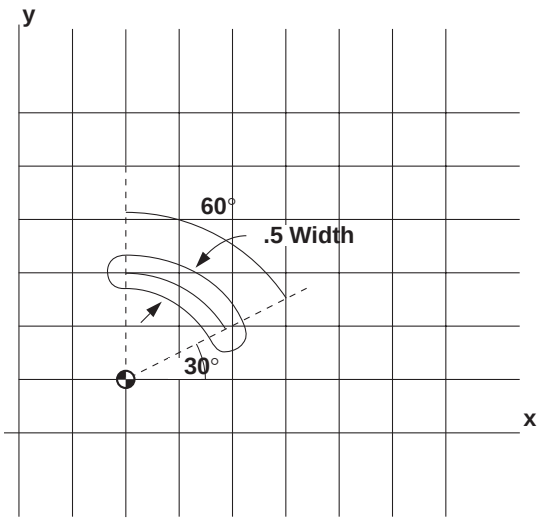


Figure 7-48

4 FACE

The **FACE** command face mills an area, designated by its length and width. In the MDI mode, this command also requires the starting position of the tool. Unlike the **FACE** command in the DO EVENT mode, the MDI FACE command does not require the tool to be at position and depth before the block is executed.

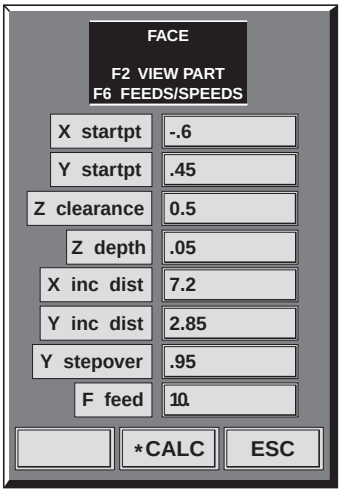


Figure 7-49

X startpt	the X coordinate of the start point for the FACE operation
Y startpt	the Y coordinate of the start point for the FACE operation
Z clearance	the clearance height above the part
Z depth	the cut depth (unsigned) (inc from clearance pt.)
X inc dist	the incremental distance to be milled along the X axis.
Y inc dist	the incremental distance to be milled along the Y axis.
Y stepover	the Y axis stepover.

For example, the programmed instruction:

```
0160 FACE|RECT X-.6 Y.45 Z.05 Z0.5
     X7.2 Y2.85 Y.95 F10.
```

will face mill the 6.0 x 3.75 block shown in Figure 7-50.

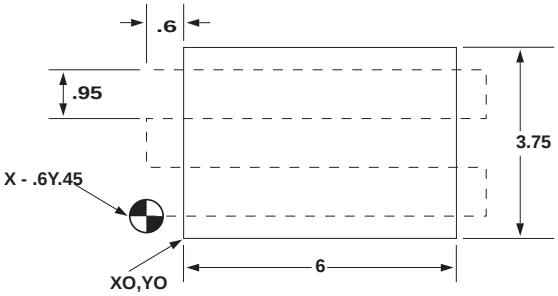


Figure 7-50

NOTE:

It is possible to start at the top right hand corner of the face area, and proceed down and to the left. In this case, the X distance, and Y distance values are entered as negatives. The Y stepover value is always entered as an unsigned distance.

+ AUXFUNC

The **AUXFUNC** (Auxiliary Function) command can be used in several different ways.

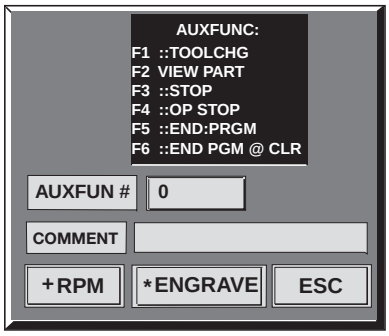


Figure 7-51

1) It is used to stop the program so that a tool change can be made. If programmed with the tool number, the control will prompt the machine operator to change to the specified tool (**F1 TOOLCHG**).

2) It is used to place a stop in the program, so that the program is halted until the operator restarts it by pressing the **START** button (**F3 STOP**).

3) It is used to place a conditional stop in the program, so that the program is halted until the operator restarts it by pressing the **START** button ONLY if the **OPTIONAL STOP** function has been set in the **RUN** mode. If the **OPTIONAL STOP** function has not been set, the conditional stop is ignored. (**F4 OP:STOP**)

4) It is used to end the program (**F5 END: PRGM** or **F6 END:PGM@CLR**).

5) It can be used to set a spindle speed for reference, so that the program may be transferred to, and executed on, another Bridgeport control (**+ RPM**). When the instruction is executed, a message will appear prompting the operator to set the programmed spindle speed.

6) It can be used to enter auxiliary functions (M-Codes) into the program, such as Coolant ON/OFF (M08, M09) so that the program may be transferred to, and executed on, another Bridgeport control.

F1 TOOLCHG To use the **AUXFUNC** command to stop the program, and prompt the operator for a tool change, press the **F1** key, and enter the number of the new tool in the TOOL field. The instruction **:: TOOLCHG T_** is placed in the program.

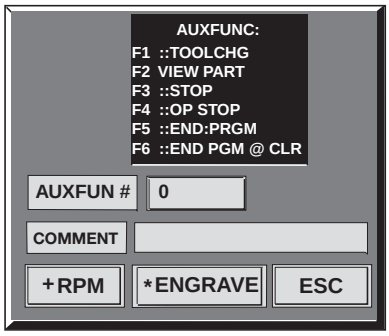


Figure 7-52

F2 VIEW PART The **F2** key in the **PGMSTOP** command will display the part preview mode. This does not enter an instruction into the program.

F3 STOP To use the **PGMSTOP** command to stop the program execution until the operator presses the **START** key again, press the **F3** key. The instruction **:: STOP** is placed in the program automatically.

F4 OP:STOP To use the **OPTIONAL STOP** command to conditionally stop the program until the operator presses the **START** key again, press the **F4** key. The instruction **::OP:STOP** is placed in the program automatically.

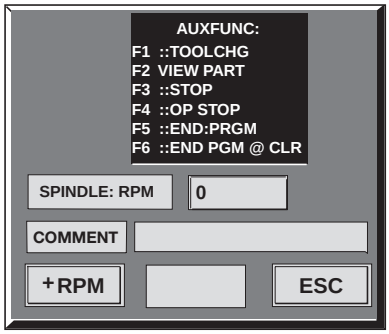


Figure 7-53

F5 END: PRGM To use the **PGMSTOP** command to end the program, press the **F5** key. The program instruction **:: END: PRGM** is automatically placed in the program.

F6 END:PRGM@CLR To use the AUXFUNC command to end the program at the clearance point, press F6. The instruction ::END:PRGM@CLR is automatically placed in the program.

+ RPM To program a Spindle Speed select the **AUXFUNC** command then press the **+ RPM** key. Enter the desired spindle speed. The instruction **:: SPINDLE ON S_** is placed in the program. When it is active, this line will prompt the operator to make an RPM change.

COMMENT: A comment line is added to [+ AUXFUNC] to allow the operator to (1) document his program, (2) prompt him to perform certain events and (3) notify him of certain events.

* ENGRAVE

The ENGRAVE command cuts a row of text in the form of letters and decimal numbers.

The command works differently in MDI than it does in DO EVENT mode. Select + AUXFUNC from the main MDI menu, then choose *ENGRAVE at the next window. The user is prompted for the engraving parameters (Fig. 7-54).

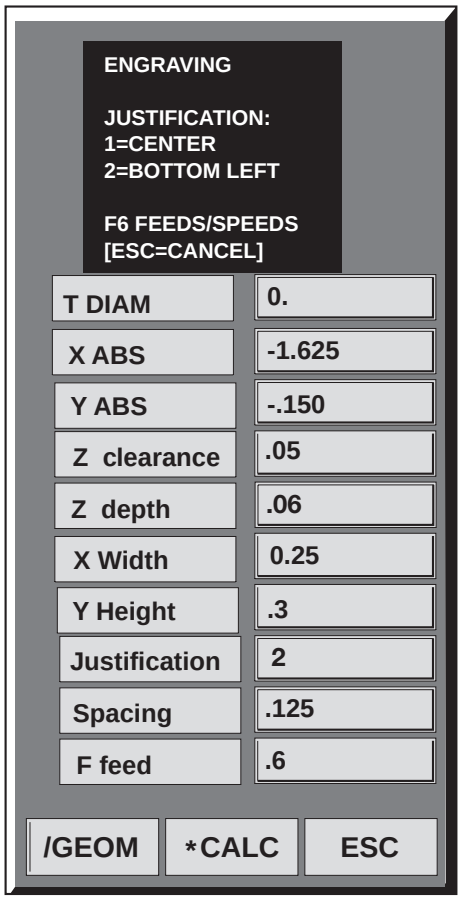


Figure 7-54

T DIAM	the tool diameter
X ABS/Y ABS	the start point for the text
Z clearance	the clearance over the part
Z Depth	the depth (unsigned) (incremental from clearpoint)
X Width	the width of each character
Y Height	the height of each character
Justification (1,2)	the start point location. 1 = start point is located at the center of the first character. 2 = start point is located at the bottom left of the first character.
Spacing	the distance between characters
Feed rate	the rate at which the tool is fed into the work

For example, we'll use the following text:

3 AXIS EZTRAK

each character to be .3 in. high (Y height) and .25 in wide (X width).

Valid characters are **0-9, A-Z** (upper case only), **+, -**, and **.**.

After parameters have been entered, select F4 TEXT. The following window appears.

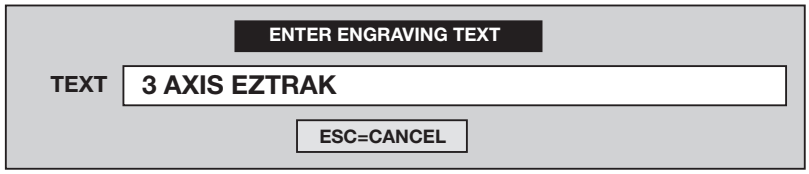


Figure 7-54

NOTE: If the host machine has the new style keyboard the text can be entered directly (Figure 7-54). If it has the old style keyboard, the user must enter *text codes* that correspond to each engraving character. See the Text Code List in the Engrave section of Chapter 5, DO EVENT.

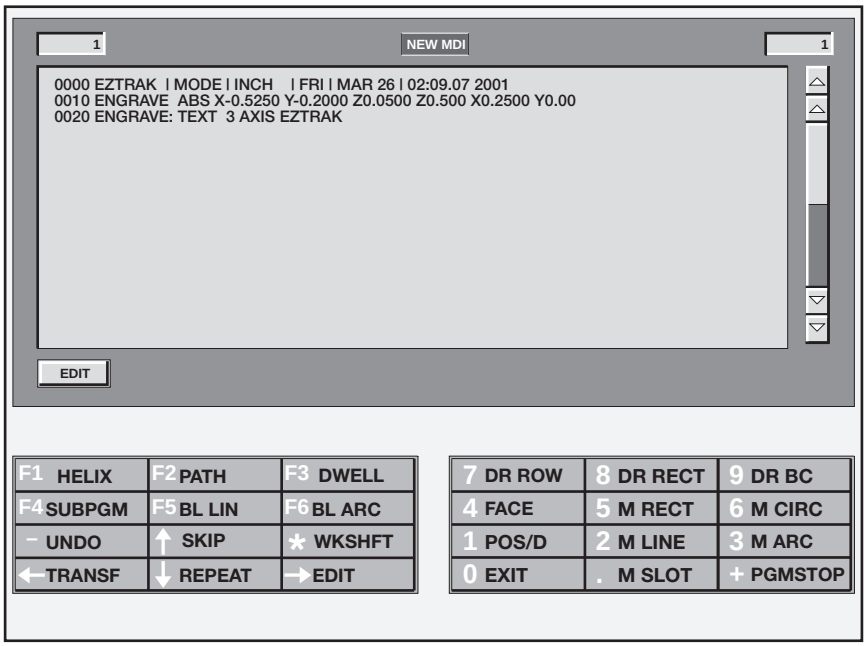


Figure 7-55

After text has been entered, you will return to the MDI main menu, where the parameters appear followed by the text. Select 0 EXIT.

F2 VIEW PART

The MDI commands have an additional feature, not found in the DO EVENT mode. Most of the MDI milling commands can display a preview of the part program geometry by pressing the **F2 VIEW PART** key while one of the instruction windows is shown on the screen. This displays a graphical preview of the part program geometry. Some program transformations (REPEAT, ROTATE, TRANSL) and the ramp-on and ramp-off moves created by the COMPON and COMPOFF commands are not displayed in this VIEW mode.

A typical **F2 VIEW PART** screen is shown below.

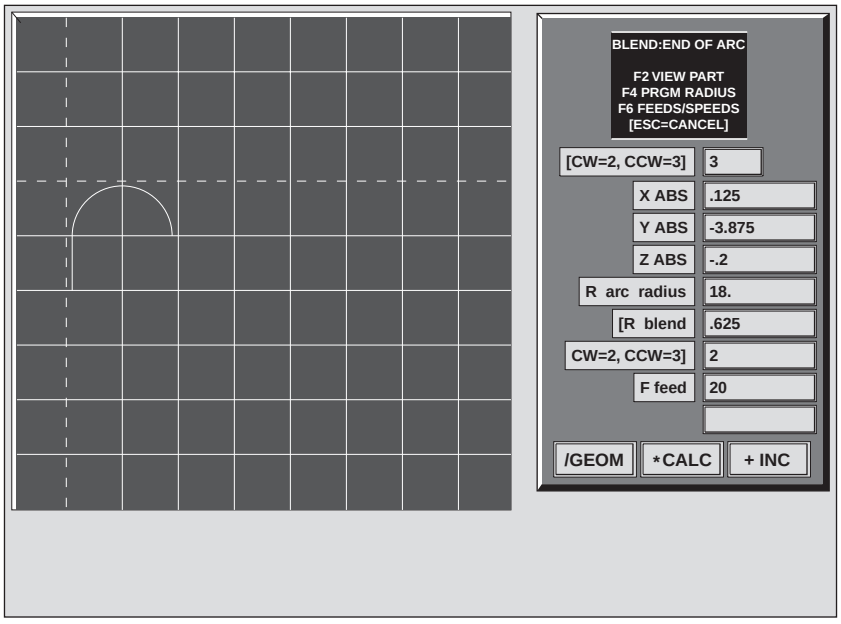


Figure 7-58

To view the actual tool path, and the complete part program, including translations, subprograms and repeats, the **PREVIEW** command in the **RUN** mode must be used. See Chapter 9 in this manual for more details on the **RUN** mode and Chapter 12 for more information on the **PREVIEW** mode. For a description of the **GRAPHIC GEOMETRY HELP** feature, see Chapter 8.

7 DR ROW:

F5 SKIP HOLES

The user can specify up to 15 hole numbers to skip in a drill cycle **(Drill Row, Drill Rectangle, Drill Bolt Circle, and/or Drill Arc)**. When specified, it will apply to all drill cycle blocks that follow the skip holes block until changed by another skip holes block.

To clear skip holes, enter in all zeroes.

For example, to drill a row of 4 holes 0.5 inch deep starting at X0 Y0 and ending at X3 Y0 but skip the 1st and 3rd holes, enter in the following PGM blocks.

```
0010 SKIP|HOLES P1 P3 P0 P0 P0 P0 P0 P0 P0 P0 P0 P0 P0 P0 P0
0020 DR|ROW ABS X0 Y0 Z0.1 Z0.5 Z0.5 Z0.5 X3. Y0. P4. F30.
```

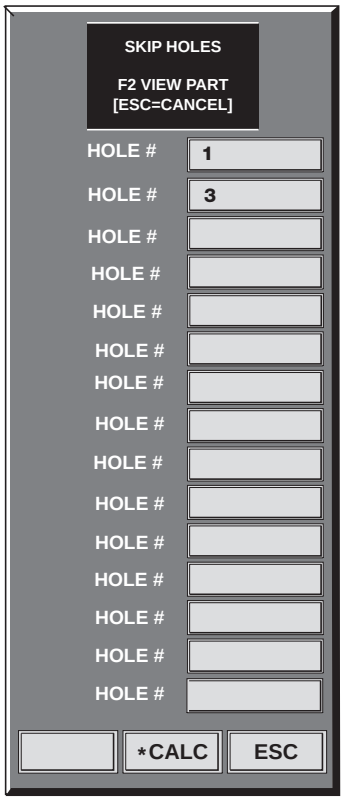


Figure 7-59

PECK CLEARANCE

A Z axis peck clearance is added to drilling canned cycles to cut down on tool wear and breakage.

When drilling a hole, the Z axis typically “pecks,” or rapids up to clear chips from the hole, then down again to drill. Tool wear and breakage can result if the tool repeatedly strikes metal with the rapid downward force of the Z axis. A peck clearance point is established so that the Z axis will pause above the hole, then feed downard from that point, lessening impact on the tool.

Figure 7-60

To set a peck clearance as a default in the control:

1. In BASIC Menu, select * (hidden key).

2. A window appears: ENTER KEYWORD. Type 01 then Enter.

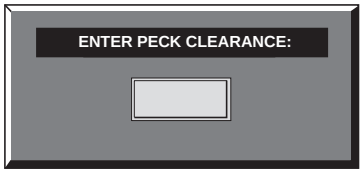


Figure 7-61

3. At the “Enter Peck Clearance” window, type in the value you wish for the clearance between pecks, then press Enter.

4. A message will appear briefly at the top of the screen: “”PECK CLR SET TO (your value).” This will be the default peck clearance for all drilling commands until it is changed.

Chapter 8
EDIT MODE

8.1 EDITING PROGRAMS

The EZTRAK has a dedicated part program editor which is called from the BASIC OPERATIONS screen by pressing the **5 EDIT** key. The editor is capable of recognizing the format of each command in the part program. When a line is selected for editing, it is displayed in the data fields used to program the line in MDI programming mode.

When **5 EDIT** is selected from the BASIC OPERATIONS screen, the screen displays a list of files that are currently available for editing, as shown in Figure 8-1 .Files are listed by format-- **Conversational (PGM), G-Code (TXT),** and **DXF**-- and you can switch among formats using function keys described in the upper portion of the window.

The ← and → arrow keys page up or down the list. Use the ↑ and ↓ arrow keys to move the highlight bar to the desired file. When the file you want is highlighted, press the + key to select the file and load it into the Editor.

Pressing the *** TOOLS** key will list the tool table files. See Section 8.5 (Tools) for more information.

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

The EDITOR also provides specific commands for inserting and deleting lines, renumbering the program instructions, and controlling the line numbers in the part program. Each command available in the EDITOR is listed in this chapter.

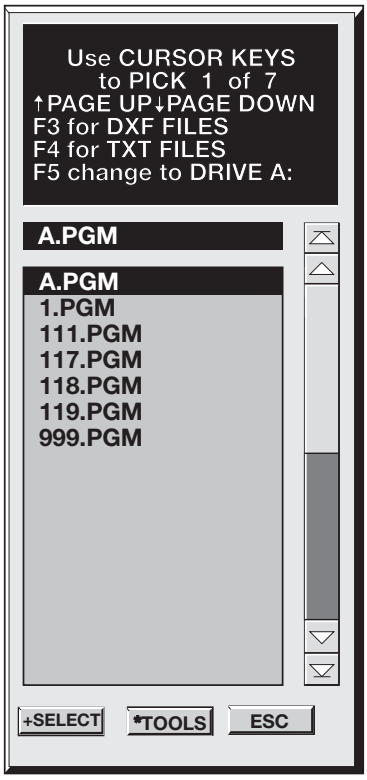


Figure 8-1

NOTE: The EZTRAK always remembers the last loaded file. The file selection will come up with the last loaded file highlighted.

8.2 EDIT MODE: CONVERSATIONAL

Once a PGM file is selected, the screen below appears, with keys available for editing the program.

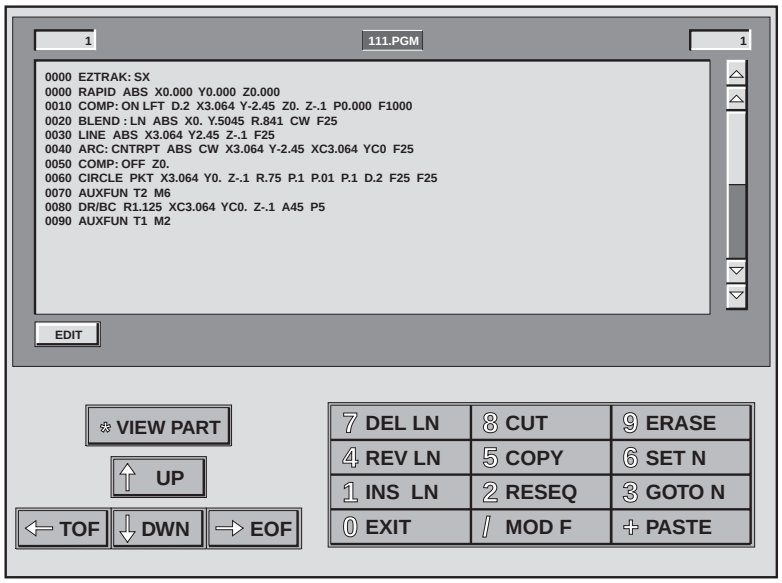


Figure 8-2

9 ERASE The **ERASE** command is used to delete a group of lines at one time. The screen displays prompts for **BEGIN at SEQNO** (the number of the first line to be deleted) and **END at SEQNO** (the number of the last line to be deleted).

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

7 DEL LN This command deletes the line on which the cursor currently appears. It is a good idea to use the **2 RESEQ** command after deleting one or more lines from a part program. **You cannot delete the first line of the part program.**

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

5 COPY The **COPY** command is used to copy a group of lines into a temporary buffer named COPY.TMP. The screen displays prompts for **BEGIN at SEQNO** (the number of the first line to be copied) and **END at SEQNO** (the line number of the last line to be copied). **COPY** is use in conjunction with the **PASTE** command. NOTE that the buffer COPY.TMP will contain the last text copied such that it can be used as a "clipboard" - to paste data from one program into another.

4 REV LN This command is used to revise any line in the part program. Using the cursor arrow keys, place the cursor on the line to be revised, then press the **4 REV LN** key. The line is read and broken down into the data fields that were used to create the operation. The data in each field of the command can be changed with a few exceptions. The nature of a command cannot be changed, e.g. a line with the M ARC operation cannot be changed to M LINE. This must be changed by deleting the offending line and then inserting a new line. ***NOTE: The first line in the part program, 0000 EZTRAK 1 MODE\INCH cannot be edited.***

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

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

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

/ MOD F The / key in the EDITOR is used to modify the programmed feedrate in a group of lines. The screen displays prompts for **BEGIN at SEQNO**, **END at SEQNO**, and **modify F to**. All of the programmed feedrates in the text from the **BEGIN at SEQNO** and up to and including the **END at SEQNO** line, will be changed to the modified feedrate value.

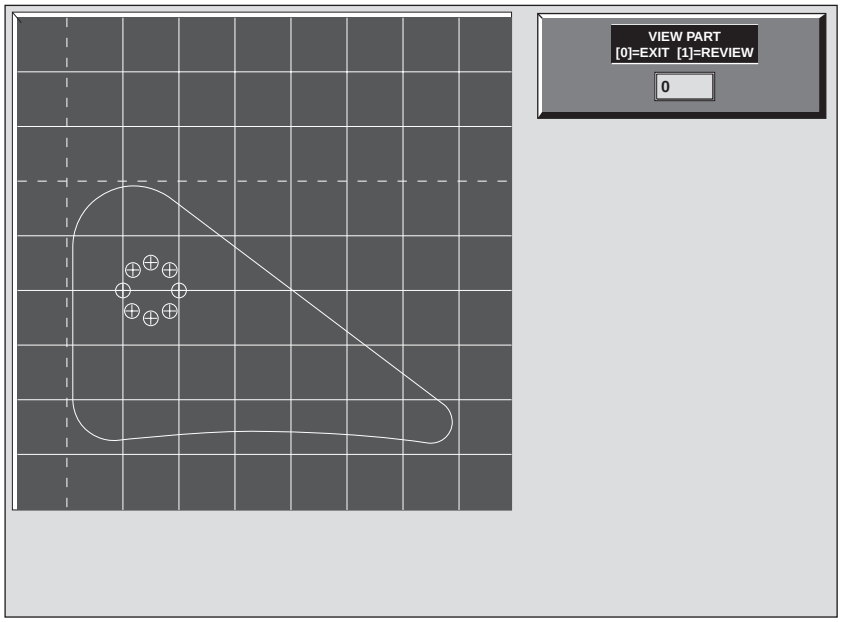


Figure 8-3

+ PASTE This command inserts the text from the temporary buffer COPY.TMP into the program **after** the current line. The buffer COPY.TMP is **not** destroyed after the text is pasted, so that the text may be pasted multiple times and in multiple programs. See the CUT and COPY commands to insert text into the COPY.TMP buffer.

***VIEW PART** The **VIEW PART** command displays the part program geometry on the screen (Figure 8-3). The operator is prompted for the first line number to be viewed, and the last number to be viewed. When the part is viewed, a prompt appears to either **EXIT** the VIEW PART mode, or **REVIEW**. The **REVIEW** command allows the user to enter new line numbers to view the part again.

< → > **EOF** This command moves the cursor to the last line of the part program.

< ← > **TOF** This command moves the cursor to the first line of the part program. This is useful for positioning the cursor before using the **GOTO N** command.

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

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

0 EXIT The 0 key in the EDIT mode exits the Editor and prompts the user to save the edited program. A window will then appear (Figure 8-4) prompting you to save the information to disk **(+ SAVE),** save to disk and load into memory to use immediately **(*SAVE:RUN),** or save to disk, load to memory, and preview the program **(/VIEW PATH)**. The name can be changed at this point so that a new file is created, leaving the original file unchanged. The new program name can be typed in place of the existing name in the PRGM field. The file extension .PGM is added automatically.

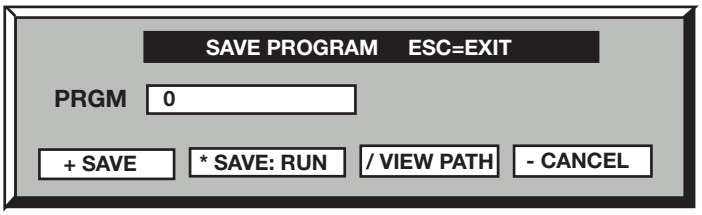


Figure 8-4

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

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

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

/ VIEW PATH This command saves the program, exits the EDIT mode, and displays the VIEW mode. The program can be previewed by using the commands described in Chapter 12.

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

8.3 EDIT MODE: G-CODE

The EZTRAK supports full G-Code programming and editing. When a G-code program is selected from the file selection box in EDIT, the MS-DOS® Editor is launched and the selected G-Code file can be edited. A G-Code Reference Guide is at the end of this section, but for instructions on how to program in G-Code, you will need a Bridgeport BPC programming manual. **NOTE: G-Code can be programmed and edited only on the new flat panel control. Old style EZTRAK controls can load and run G-code programs but not edit or program them.**

Keyboard

The new keyboard was designed so that the most commonly used characters can be easily accessed. The keyboard contains all of the letters of the alphabet, all of the punctuation characters, and all of the necessary editing functions. Most keys have two functions. The character in larger print will appear when its key is pressed. To type the character in the upper left corner, hold down the Shift key then press the desired key. All of the characters in Bridgeport G-Code language and PPPL (Parametric Part Programming Language) are available.

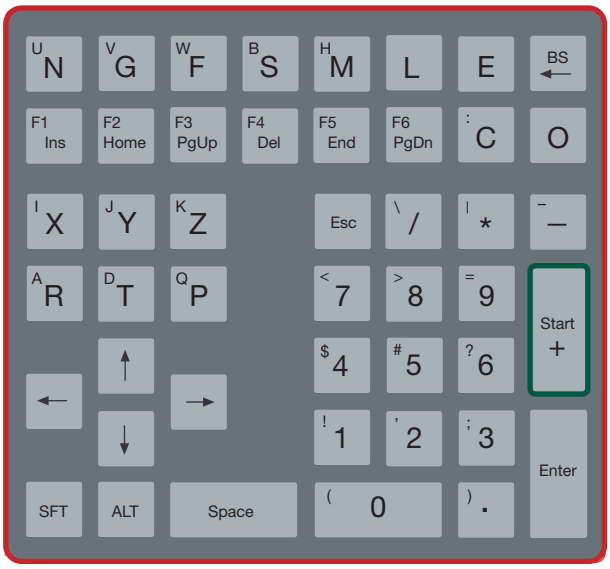


Figure 8-5

Accessing the Editor

From the Main Screen press [5 EDIT] and select [F4 MORE FILES]. The G-code files will appear in the file selection box. Select the desired file by pressing the Up arrow, Down arrow, Page Up, or Page Down keys, then press [+ SELECT]. The following message appears before Edit is launched.

REMEMBER TO RELOAD G-CODE FILE AFTER EDITING.
PRESS ANY KEY TO EDIT G-CODE OR ESC TO CANCEL.

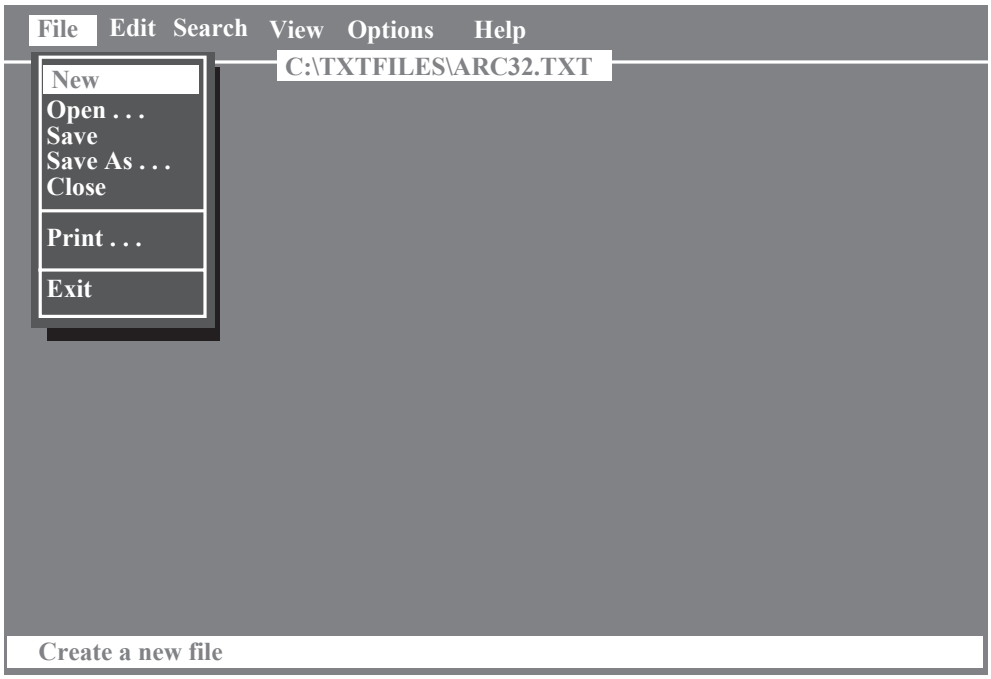


Figure 8-6

Creating A New File

To create a new file, open an existing G-Code file. Press the ALT key to highlight “File” on the Menu Bar then press Enter. Select ‘New” to create a new file, and press Enter again A new edit screen will appear...The word UNTITLED1 will appear in the file name area in the center top of the screen (SHORT CUT: ALT FN). Type in your G-Code program. When you have finished creating the G-Code program, press the ALT key, followed by the ENTER key followed by the “A” key [SAVE AS]. Enter the name of the new file, then press ENTER.

Exiting The Editor

Press and release the ALT key, press the Enter key, then press the ↓ key until the word EXIT is highlighted, then press the Enter key. (SHORT CUT: ALT FX). If you have made edits to the file, a message stating that “the file has not been saved yet. Save it now?” will appear. If you wish to save your edits press the Enter key. If you do not wish to save your edits, press the ➔ key then press Enter. If you wish to continue editing the file, press the ➔ key until the cursor is under the text “Cancel”, then press the Enter key. When you exit the Editor, you will be returned to the EZTRAK Main Screen.

To load the file into memory to be executed, select [+ RUN], [←LOAD], and [F4 MORE FILES]. Select the desired file and press [+ LOAD].

Most Commonly Used Editing Functions

The most commonly used editing functions are described below. To obtain more complete detailed information, when in the editor hold down the Shift [SFT] key then press the F1 key to access the built in Help utility.

The flashing underscore is called the cursor. The cursor location is where the next typed character will appear.

Moving the cursor

Key	*Function*
Up Arrow Key	Moves the cursor up one row.
Down Arrow Key	Moves the cursor down one row.
Left Arrow Key	Moves the cursor to the left one column.
Right Arrow Key	Moves the cursor to the right one column.
Home Key	Moves the cursor to column 1
End Key	Moves the cursor to the end of the line of text
PgUp Key	Moves the cursor up 22 rows
PgDn Key	Moves the cursor down 22 rows

Manipulating Text

Key	*Function*
BS Key	Backspace function. Moves the character that is above the cursor to the left one column thereby deleting the character at that location.
INS Key	Toggles between INSERT and OVERSTRIKE mode. In Insert mode, the cursor appears as a blinking underscore character. In overstrike mode, the cursor becomes a flashing rectangle. In Insert mode, typed characters are inserted where the cursor appears. In overstrike mode, the typed character overwrites the character at the current cursor location.
DEL Key	Deletes the character at the current cursor location

Selecting Text

Press and hold down the **SFT** key then press one of the following 4 arrow keys.

Key	*Function*
Left Arrow	Highlights the character at the current cursor location and moves the cursor 1 column to the right.
Right Arrow	Highlights the character at the current cursor location and moves the cursor 1 column to the left.
Up Arrow	Highlights the row that the cursor is on.
Down Arrow	Highlights the row that the cursor is on and moves the cursor down one line..

Editing Selected Text

Once text is selected, the functions Cut, Copy, and Paste can be performed on the selected text.

Cut Removes the selected text and stores it in the paste buffer.
Press and unpress the ALT key, press the right arrow key and the word EDIT is highlighted. Press the Enter key 2 times. (SHORT CUT: ALT ET)

Copy Stores the selected text in the paste buffer.
Press and unpress the ALT key, then press the right arrow key and the word EDIT is highlighted. Press the Enter key then press the down arrow key until the word COPY is blackened then press the Enter key.
(SHORT CUT: ALT EC)

Paste Copies the selected text in the paste buffer to the editing area at the current cursor location. Press and unpress the ALT key, then press the right arrow key and the word EDIT is highlighted. Press the Enter key then press the down arrow key until the word PASTE is hihglighted then press the Enter key. (SHORT CUT: ALT EP) You can paste the same text to another location by moving the cursor to that desired location and then repeat the above procedure.

Clear Press ALT and EE to clear.

Searching For Text

Press and release the ALT key. Press the ➔ key until the word SEARCH is highlighted. Press Enter two times, then type in the desired text and press the Enter again (SHORT CUT: ALT SF). If the desired text exists, the cursor will move to the start of the first occurrence.. If the desired text does not exist, the message "Edit was unable to find a match" will appear. Press the Enter key to continue. If the desired text was found, you can find the next occurrence by pressing the F3 key. Note that Edit always searches *forward* starting from the current cursor location to the end of the file. So if the text you wish to find is above the cursor, it will not be found. You need to move the cursor to the top of the file to begin the search.

Replacing Text

Press and release the ALT key. Press the right arrow key until the word SEARCH is highlighted. Press Enter then press the ↓ key until the word REPLACE is highlighted (SHORT CUT: ALT SR). Press the Enter key and type in the text you wish to find next to the FIND WHAT prompt. Press the ↓ key and type in the text to replace it with next to the REPLACE WITH prompt. Press the ↓ key until the cursor appears under the Replace button. If you wish the editor to prompt you at each occurrence, just press the Enter key. If you wish to replace all occurrences without any prompting, press the ➔ key until the cursor appears under the REPLACE ALL button, then press Enter. Note that Edit always searches *forward* starting from the current cursor location to the end of the file. So if the text you wish to find is above the cursor, it will not be found. You need to move the cursor to the top of the file then do the replace.

programming code SUMMARY for mills
G-Code (Preparatory) & M-CODE (Miscellaneous)

G-Code	preparatory Function
	MOTION
*G0	Rapid Traverse
G1	XYZ Linear Interpolation (Feed)
G2	Circular Interpolation, CW
G3	Circular Interpolation, CCW
G4	*Dwell*
G12	*Helical Interpolation, CW*
G13	*Helical Interpolation, CCW*
G22	*Circular Interpolation, Fillet Input CW*
G23	*Circular Interpolation, Fillet Input CCW*
*G8	Deceleration Override OFF
G9	Deceleration Override ON
G99	*Deceleration Override, Single Block*
G74	Single Quadrant Circle Input ON
*G75	Multi-Quadrant Circle Input ON
	PLANE SELECTION
*G17	XY, Plane Designation
G18	XZ, Plane Designation
G19	YZ, Plane Designation
	TOOL RADIUS COMPENSATION
*G40	Cutter Comp. OFF/CANCEL
G41	Cutter Comp. LEFT, Left of Contour
G42	Cutter Comp. RIGHT, Right of Contour
*G44	Cutter Comp. Normal Feedrate
G45	Cutter Comp. Constant SFM
G48	Corner Rounding with Cutter Comp. OFF
*G49	Corner Rounding with Cutter Comp. ON
	COORDINATE TRANSFORMATIONS
*G30	Mirror Image OFF/CANCEL
G31	Mirror Image X
G32	Mirror Image Y
*G72	Transformation OFF/CANCEL
G73	Transformation, Scaling or Rotation
	WORKSHIFTS
*G54	Default Fixture Offset UCS
G55	Additional Fixture Offset
G56	Additional Fixture Offset
G57	Additional Fixture Offset
G58	Additional Fixture Offset
G59	Additional Fixture Offset
G92	*Permanent Datum Shift*
*G96	CANCEL G97
G97	Temporary Datum Shift

G-Code	preparatory Function
	DIMENSIONS
G70	Input in Inches
G71	Input in Millimeters
*G90	Absolute Programming
G91	Incremental Programming
	FEEDRATE
*G94	Feedrate Per Minute Mode
G95	Feedrate Per Revolution (pitch) Mode
	DRILLING CYCLES
*G80	Drill Cycle OFF/CANCEL
G81	Drill (Feed In, Rapid Out)
G82	Spot Face (Feed In, Dwell, Rapid Out)
G83	Deep Hole (Peck, Rapid Out)
G84	Tap (Feed In, Feed Out)
G85	Bore (Feed In, Feed Out)
G86	Bore (Feed In, Orient Spindle, Rapid Out)
G87	Chip Break (Peck, Rapid Out)
G89	Bore (Feed In, Dwell, Feed Out)
	MILLING CYCLES
G77	*Zig-Zag*
G78	*Rectangular Pocket*
G79	*Circular Bore*
G170	*Outside Frame*
G171	*Inside Frame*
G172	*Pocket Frame*
G173	*Outside Face*
G174	*Inside Face*
G175	*Outside Circle*
G176	*Inside Circle*
G177	*Pocket Circle*
G179	*Slot*
	Z-AXIS MULTI-HOLE ROW DRILLING CYCLES
*G180	Drill Cycle OFF/CANCEL
G181	Drill (Feed In, Rapid Out)
G182	Spot Face (Feed In, Dwell, Rapid Out)
G183	Deep Hole (Peck, Rapid Out)
G184	Tap (Feed In, Feed Out)
G185	Bore (Feed In, Feed Out)
G186	Bore (Feed In, Orient Spindle, Rapid Out)
G187	Chip Break (Peck, Rapid Out)
G189	Bore (Feed In, Dwell, Feed Out)
	Z-AXIS FRAME OF HOLES DRILLING CYCLES
*G190	Drill Cycle OFF/CANCEL
G191	Drill (Feed In, Rapid Out)
G192	Spot Face (Feed In, Dwell, Rapid Out)
G193	Deep Hole (Peck, Rapid Out)
G194	Tap (Feed In, Feed Out)
G195	Bore (Feed In, Feed Out)
G196	Bore (Feed In, Orient Spindle, Rapid Out)
G197	Chip Break (Peck, Rapid Out)
G199	Bore (Feed In, Dwell, Feed Out)

programming code SUMMARY for mills (cONTINUED)
G-Code (Preparatory) & M-CODE (Miscellaneous)

G-Code preparatory Function

DIGITIZING CYCLES (OPTIONAL SOFTWARE)

*G200	Digitizing Cycle OFF/CANCEL
G201	Basic - Digitize Points, Lines, Arcs
G202	Mesh - Digitize XY Mesh
G203	Z-Trace - Trace Z Normal
G204	XY-Trace - Trace XY Boundary
G205	Cross-Section - (Trace)

M-Code miscellaneous Function

M0	*Program STOP*
M1	*Optional Program STOP*
M2	*Program STOP & RESET (i.e., End of Program, RESET to Top of Program)*
M3	Turn Spindle ON CW
M4	Turn Spindle ON CCW
M5	Turn Spindle OFF (Stop)
M6	*Tool Change*
M7	Coolant, Mist (also turns ON the 'Thru the Tool Coolant' Option, if installed)
M8	Coolant, Flood
M9	Coolant, OFF
M19	*Orient Spindle (VMC Only)*
M20	*Program STOP, GO to Clear Point*
M21	*Optional Program STOP, Go to Clear Point*
M22	*End of Program, RESET to Top of Program but First Go to Clear Point*
M25	*Z-Axis Home*
M26	*Tool Change, Go to Clear Point*
M28	Cancel Rigid Tapping Mode (i.e., Puts System in Open Loop Mode, VMC's Only)
M29	Enable Rigid Tapping Mode (VMC's Only)
M30	*End of Program, RESET to Top of Current Numbered Program Only (Useful with BOSS 8, 9 & 10)*
M36	Chip Conveyor FORWARD (Optional Part, VMC's Only)
M37	Chip Conveyor OFF (Optional Part, VMC's Only)
M38	Chip Conveyor REVERSE (Optional Part, VMC's Only)
M51	*Advance Index Table*

NOTES:

This table was prepared using current Bridgeport Machine's Controls available as of 4/98.

Modal & Non-Modal are defined as follows:

- Modal type code functions will remain active or effective until another code from the same group is executed.
- *Non-Modal* type code functions are only active in the block of programming in which it is specified.

G-Code & M-Code functions are separated into groups of Modal & Non-Modal code types. Modal code functions are shown in a regular font. Non-Modal code functions are shown in an *Italic* font.

*	Denotes those G-Codes which are system defaults when the machine is first powered up or reset.
CW	Clockwise Direction
CCW	Counter-Clockwise Direction
UCS	User Coordinate System

REMEMBER:

8.4 EDIT MODE: DXF

Two-dimensional DXF files may be imported to the EZTRAK control by way of floppy disk (A drive) or RS232 cable. The EZTRAK control translates the drawings into point locations connected by lines (for example, Figure 8-7) defined as *features a*nd *segments*.

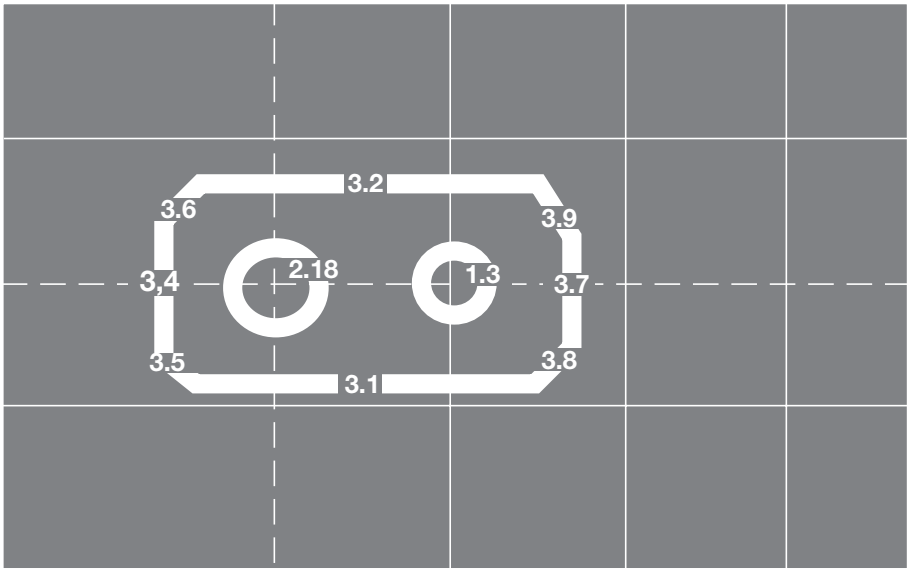


Figure 8-7

Features and Segments

Each feature is made up of a series of connected lines or arcs that create a closed shape. The individual lines or arcs that make up a feature are called segments. Each segment is tagged with a feature number followed by a segment number. There are three features in Figure 8-7. The small circle is feature 1; the other circle is feature 2; the closed shape around the circles is defined as feature 3. Feature 3 has 8 segments. Segment numbers are frequently not in a logical sequence because they simply indicate the order in which the segments were drawn.

NOTE: DXF files do not show dimensions, therefore it is necessary to have a dimensioned drawing available for specific information while writing the program.

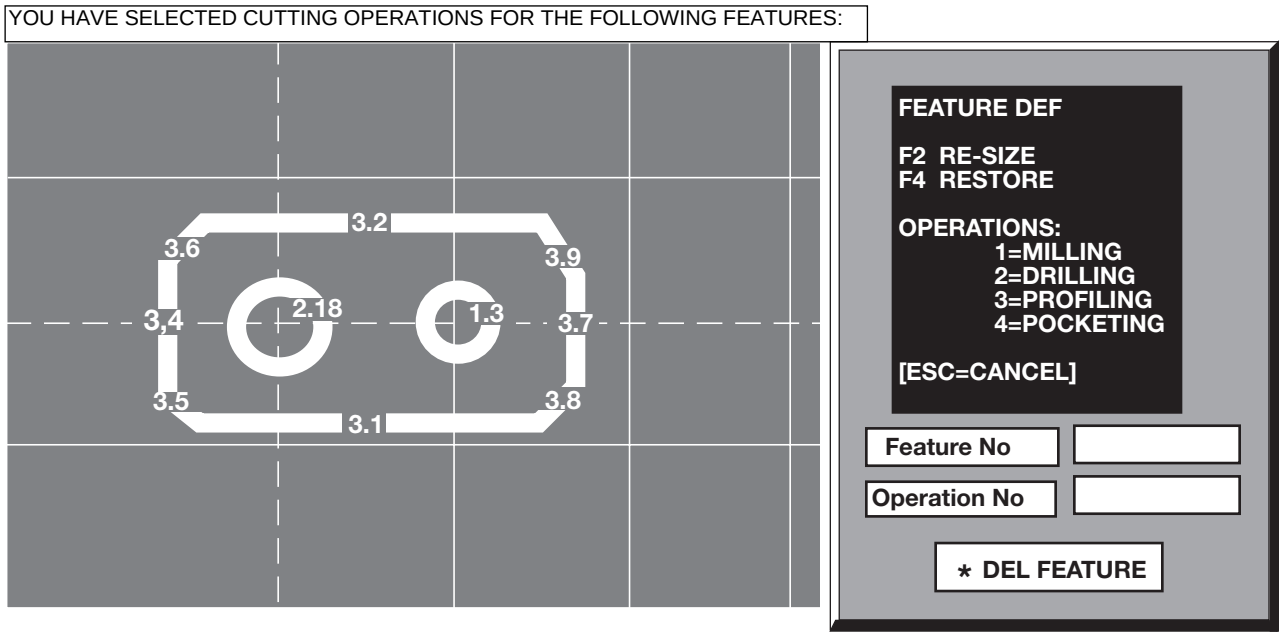


Figure 8-8

Creating and Editing DXF Files

From the 5 EDIT key select F3 DXF Files. Select the desired file to write into a program.
If the selected file has been previously edited, a message will appear indicating the features that have already been selected. To clear features that were selected, choose [*DEL FEATURE] and enter the feature number to be deleted.

NOTE: Segment ambiguity can ocur when an imported DXF file has segments that are grouped into open and closed paths, and when lines are not connective. Therefore it is necessary for all imported files to be drawn correctly..The following rules apply:

1. **Lines must be connective**
2. **Drawings must be to scale (1/1)**
3. **Lines should not overlap.**
4. **Layered drawings are not allowed.**
5. **Lines that do not apply to the view, such as hidden lines, are not allowable.**
6. **Drawings must be 2-dimensional.**

Segment Ambiguity

When a point contains 3 or more segments the direction of travel cannot be determined without user intervention. For example, in Figure 8-9, travel starts at Segment 20, then goes to 21, but after 21 the next segment can be either 22 or 24. It is up to the user to resolve this ambiguity. The segment and its ambiguous segments are highlighted and tagged and the user is prompted for the correct segment number. Note: the user can "zoom in" on the segment by selecting F2 RESIZE; to restore the start view, select F4 RESTORE.

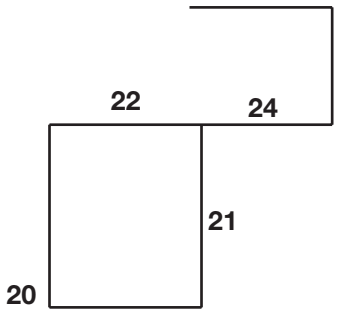


Figure 8-9

Features and Cutting Operations

When a feature number is selected, all of its segments are highlighted. After the feature has been selected, the user is prompted for the type of cutting operation to be performed on that feature. Valid cutting oerpations are **Milling, Drilling, Profiling,** and **Pocketing.**

1. MILLING

No tool radius compensation is applied to the cutting path. The tool cuts along the center line of the part. If cutter compensation is desired, select the *profiling* operation.

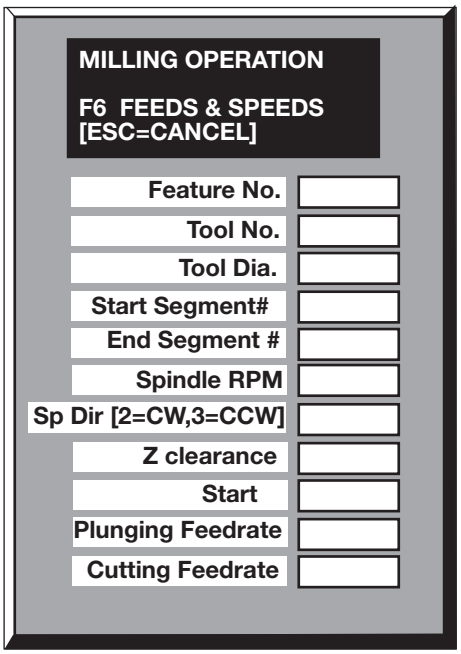


Figure 8-10

Feature Number Feature identification number. Segments comprising this feature are highlighted.

Tool Number Tool identification number. When it is time to change to this tool, a message is displayed, prompting for the tool number and tool diameter.

Tool Diameter Unsigned tool diameter. Be sure this matches what is in the tool table. Tool radius compensation is NOT applied to the cutting path.

Start Segment # The number of the first segment where cutting begins. The starting segment must be part of the selected feature. If the feature is an open path, the start segment must be open ended.

End Segment # The segment number where cutting ends. The end segment must be part of the selected feature. If the feature is an open ended path, the end segment must be open ended. If the feature is a closed path, the end segment must be connected to the start segment. Specifying the end segment determines the cutting direction.

Spindle RPM Rotational spindle velocity (Revolutions Per Minute).

Sp Dir [2=CW, 3=CCW] Spindle turning direction.

Z Clearance The signed tool clearance above the part.

Z ABS The signed milling Z position.

Plunging Feedrate Feedrate from Z clearance to Z ABS position.

Cutting Feedrate The milling feedrate.

2. DRILLING

Feature Number Feature identification number. The holes comprising this feature are highlighted.

Tool Number Tool identification number. When it is time to change this tool, a message is displayed prompting for the tool number and tool diameter.

Tool Diameter Unsigned tool diameter. This information must match what is in the tool table.

Spindle RPM Rotation spindle velocity in Revolutions Per Minute (RPM)

Sp Dir [2=CW, 3=CCW] Spindle turning direction.

Z Clearance The signed tool clearance above the part.

Z ABS The signed Z position of the bottom of the hole.

Z Peck The first pecking depth (unsigned) for the hole. (If this value equals zero or is more than the depth, it will be set to the Z depth).

Z Peck [2nd,...] The second and subsequent pecking depth (unsigned) for the hole. (If this value equals zero or is more then the depth, it will be set to the Z depth) .

Z feedrate The speed at which the tool moves into the part.

Dwell (sec) (Applies to Boring and Tapping only) The number of seconds to dwell at the top and bottom of the hole.

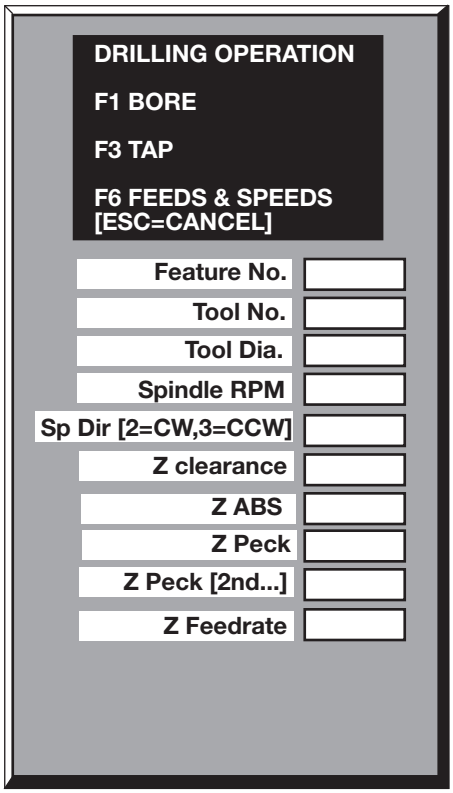


Figure 8-11

3. PROFILING

Feature Number Feature identification number. The segments comprising this feature are highlighted.

Tool Number Tool identification number. When it is time to change to this tool, a message is displayed, prompting for the tool number and tool diameter.

Tool Diameter This information is used in the message prompt at tool change and during tool radius compensation. Be sure what is entered here matches what is in the tool table.

Start Segment Number of the first segment where cutting begins. The segment must be part of the selected feature. If the feature is an open path, the start segment must be open ended.

End Segment Segment number where cutting ends. The End segment must be part of the selected feature. If the feature is an open ended path, the End segment must be open ended. If the feature is a closed path, the End segment must be connected to the Start segment. Specifying the end segment determines the cutting direction.

Spindle RPM Rotation spindle velocity (Revolutions Per Minute)

Sp Dir [2=CW, 3=CCW] Spindle turning direction.

[LFT=1, RGT=2] The direction of the cutter offset.

[approach] The approach distance.

Z approach The approach depth.

Z Clearance The signed tool clearance above the part.

Z Mill ABS The signed milling Z position.

Feedrate The milling feedrate.

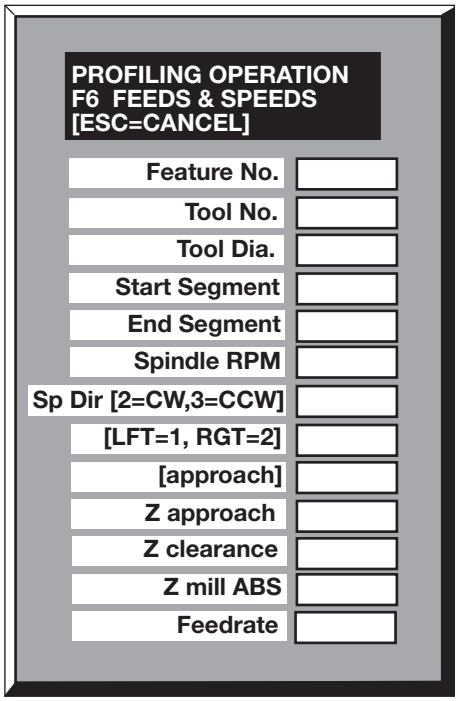


Figure 8-12

4. POCKETING

Feature Number Feature identification number. The segments comprising this feature are highlighted.

Tool Number Tool identification number. When it is time to change to this tool, a message is displayed, prompting for the tool number and tool diameter.

Tool Diameter The unsigned diameter in inches or mm of the cutting tool. The cycle uses this information to insure that the tool never moves outside the part shape.Be sure what is entered for tool diameter here matches what is in the tool table.

Path Dir [2=CW, 3=CCW] Path cutting direction.

Start Segment Number of the first segment where cutting begins. The start segment must be part of the selected feature. If the feature is an open path, the Start segment must be open ended.

End Segment Segment number where cutting ends. The End segment must be part of the selected feature. If the feature is an open ended path, the End segment must be open ended. If the feature is a closed path, the End segment must be connected to the Start segment. Specifying the End segment determines the cutting direction..

Spindle RPM Rotational spindle velocity in Revolutions Per Minute.

Sp Dir [2=CW, 3=CCW] Spindle turning direction.

Zigzag direction (1=horizontal, 2=vertical) An entry of 1 for horiztonal means tha the tool steps along the perimeter of the part the step amount and then moves back and forth in the X direction until the entire interior of the part shape (less the allowance) has been cut at the current depth. An entry of 2 (vertical) means that the tool cuts back and forth in the Y direction

Pocket Stepover The unsigned stepover amount in inches or mm, in the vertical or horizontal direction, that the tool will move before making a cutting pass the entire length or width of the part.

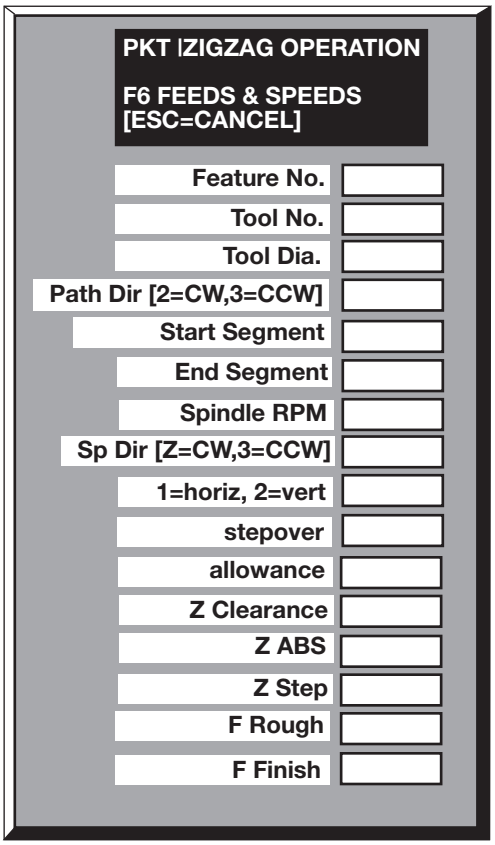


Figure 8-13

Allowance The amount of stock in inches or mm to leave along the inside perimeter during the roughing passes to be removed later during the finish pass.

Z clearance The safe ABS rapid point position. The cutter will rapid to this position before moving in the X or Y direction to the start of the next segment and at the start and end of each Z step, profile, and finish pass.

Z ABS The signed final Z position of the pocket floor for the finish pass and after the last roughing pass has been completed.

Z step The unsigned amount to lower the cutting tool before starting each roughing pass. Feeding and roughing passes are repeated until the current Z position reaches Z ABS.

F Rough Feedrate used for the profile and roughing passes.

F Finish The finish pass feedrate.

DEL Feature

This key deletes all the PGM statements associated with a feature from the part program. The user is prompted for the number of the feature that is to be deleted. All feature numbers that have cutting operations associated with them are listed and all of the segments comprising those features are highlighted.

Saving and Loading

Pressing the Escape key while the Feature Definition box is visible exits the DXF File Editor and prompts the user to save the cutting operation(s) and parameters of the selected features. The name can be changed at this point so that a new PGM file is created, leaving the original file unchanged. The new program name can be typed in place of the existing name in the PRGM field. The file extension PGM is added automatically.

8.5 * TOOLS

The **TOOLS** file is accessed from 5EDIT, as shown in Figure 8-14, by pressing the asterisk (*) key. Once this key is pressed, the user is presented with a list of tool library files to choose from (Figure 8-15). After selecting the desired tool library file, the screen shown in Figure 8-16 appears.

The **Tool Library** stores such tool information as Tool Number (**TNO**), Tool Length Offset (**TLO**), and Diameter (**DIA**) for up to 24 different tools. The library also displays Workshift coordinates **(WS)** which can be accessed and edited. (For more information on the Tool Library, see Appendix G of this manual.)

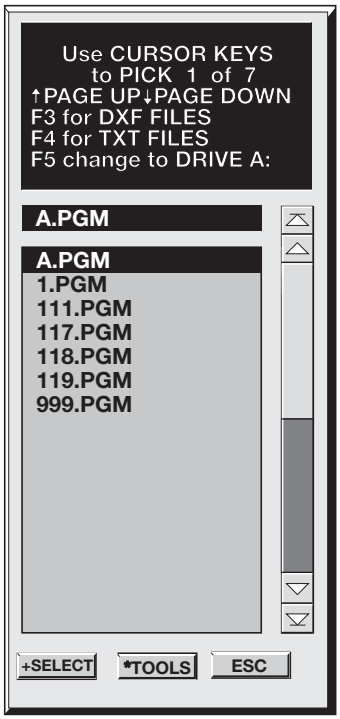


Figure 8-14

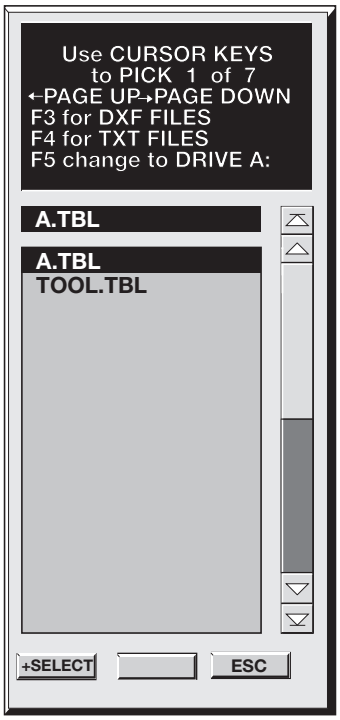


Figure 8-15

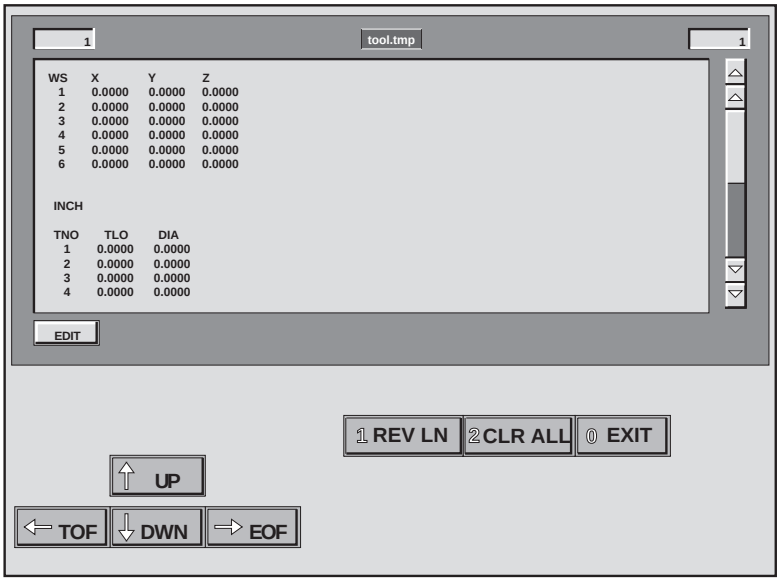


Figure 8-16

The commands available in the Tool Library are:

< ↑ > UP
< ↓ > DOWN These keys control the cursor box movement either UP or DOWN one line at a time.

< ← > Top Of File (TOF)
< → > End Of File (EOF) These keys control the cursor box movement either UP or DOWN to the beginning or end of the Tool Library file.

1 REV LN This command allows the operator to revise the data for a specific tool or a specified workshift. It is used by placing the cursor box on the line where the change is desired, then pressing the 1 key. Changes made to the tool offset value should reflect the amount of tool wear. These values can be input either in ABS, which will change the values in the table to the input values, or in INC, which will add the input values to the existing values in the table.

2 CLR ALL This command clears all the entries in the Tool Library and all workshifts by entering zero for each TLO, diameter, and workshift coordinate.

0 EXIT This command exits the Tool Library. When this key is pressed, a screen appears (Figure 8-17) and the operator is asked to save the Tool Library and workshift. At this screen the operator can choose to **+ SAVE** the tool table, **– CANCEL** which allows the operator to go back into the tool table, or **ESC = EXIT** which would bring the **BASIC OPERATIONS** screen up and NOT save any changes to the Tool Library. This screen also prompts the user for a file name. The name can be changed at this point so that a new file is created, leaving the old file unchanged. The new tool library name can be typed in place of the existing name in the Tools field. The file extension .TBL is added automatically.

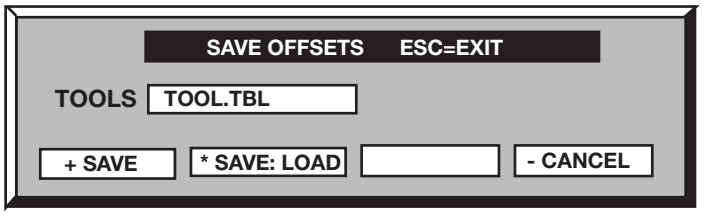


Figure 8-17

Chapter 9
RUN MODE

INTRODUCTION

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

9.1 THE RUN SCREEN

When the **RUN** command is selected, the screen displays:

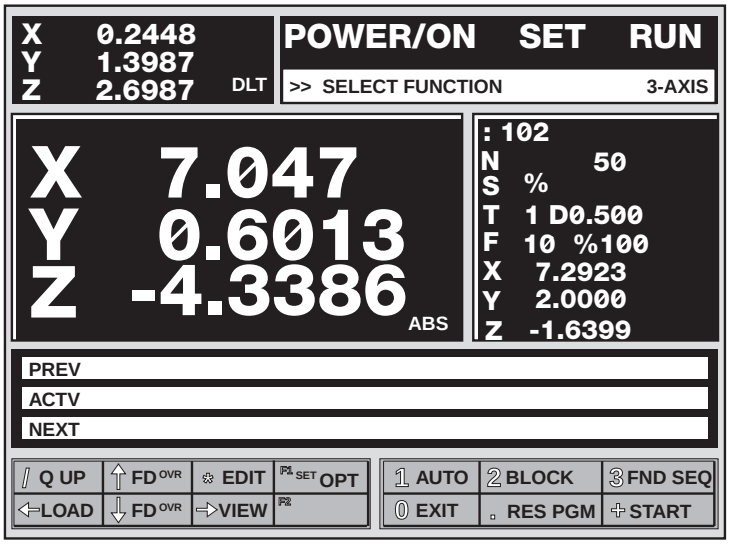


Figure 9-1

9.2 KEY FUNCTIONS

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

1 AUTO The **AUTO** command sets the loaded program to be run in continuous operation. The program stops for Z axis adjustment (in 2-axis mode), for programmed stops, or tool changes. The program run begins when the **START** key is pressed.

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

NOTE: Canned cycles are treated as single block instructions. Use the HOLD button on the Operator's Panel to pause execution of canned cycles.

Figure 9-2

3 FND SEQ This command searches the active part program for a specific sequence number or tool change command The **FND SEQ** command displays a window on the screen as shown in Figure 9-2.

Press the + key to find a sequence number in the active part program, or press the * key to find a tool change by the tool number. Once the sequence number or tool change is found, the program will begin execution at that line. If the sequence number or tool cannot be found, press .RES PGM to rewind to the start of the program.

Press the **ESC** key to exit to the FND SEQ mode.

. RES PGM This command is used to reset the active program back to the beginning.

+ START. The **START** command begins program execution in either the **BLOCK** mode or the **AUTO** mode. The **START** key is also used to restart the program after it has stopped for drilling, Z axis adjustment (2-axis mode only), or a programmed stop.

<↑> <↓> FDR OVR The FDOVR commands are used to override the programmed feedrate. The <up arrow> key raises the programmed feedrate by 5%. The <down arrow> key lowers the programmed feedrate by 5%. The feed override amount is shown as a percent on the right side of the screen.

<←> LOAD The **LOAD** command is used to choose a part program to run or to load a tool library file. When a program is chosen, it becomes the active program in memory.

When the **LOAD** command is selected, the file list shown in Figure 9-3 is shown on the screen. The <up arrow> and <down arrow> cursor keys can be used to scroll through the list of files. Press the **+ LOAD** key when the desired file is highlighted.

In the file list, only **.PGM** files are shown. To display a list of the G-Code **.TXT** files, press the **F4** key. The list shows only the **.TXT** files that are on the EZTRAK. To display a list of the tool library .TBL files, press the * key. To list the files that are on the 3.5 inch diskette in the disk drive, press the **F5** key. At first, only the **.PGM** files are shown. Press the **F4** key to list**.TXT** files. Press the * key to list.TBL files. Press **F3** to list DXF files. (DXF files must be converted to PGM files in order to run. See Chapter 8 - Edit Mode: DXF.)

The <←> and <→> cursor keys can also be used to move through the list of files very quickly. The <→> key advances downward through the file list one page at a time. The <←> key moves upward through the list one page at a time.

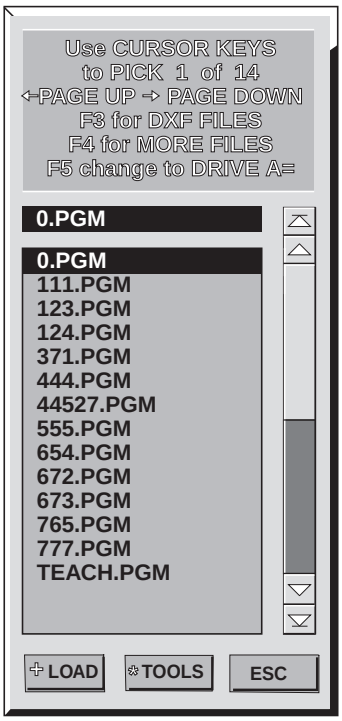


Figure 9-3

Press the **ESC** key to exit the **LOAD** command without loading a file. **NOTE: The EZTRAK always remember the last loaded file and the file selection comes up with the last loaded file highlighted.**

<→> VIEW This enables previewing the part-program by simulating the cutter path on the display screen. For more information on the **VIEW** mode, see Chapter 12 in this manual.

F1 SET OPT This command is used to set the RUN options for the program. The following screen appears when this key is pressed.

Each selection is controlled by a toggle select key. Press the desired number key of the function once and an "X" will appear in the check box, press it again and the "X" will disappear. If there is an "X" in a check box, it means the function is turned on. Any or all of the options can be active at the same time.

1 OPTIONAL STOP This function will cause any programmed OPTION STOP (M01) command to be activated, meaning that if this function is turned on, any OPTION STOP in the program will cause the program to stop execution. If this function is not turned on, then the OPTION STOP in the program will be ignored and the program will continue to run until a STOP or PROGRAM END is reached.

NOTE: OPTIONAL STOP is programmed in the MDI mode by pressing +PRGMSTOP then F4 OP STOP.

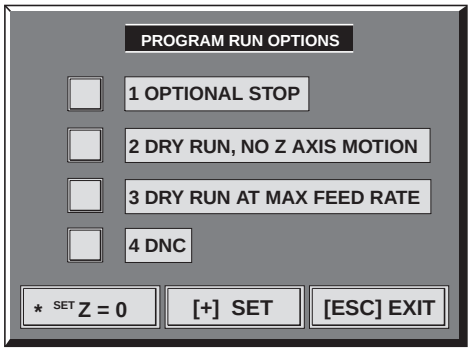


Figure 9-4

Figure 9-5

2 DRY RUN, NO Z AXIS MOTION This function will run the program at the programmed feedrate but not move the Z axis.

3 DRY RUN AT MAX FEEDRATE This function will run the program with Z axis motion at RAPID speed.

4 DNC This function will run a program in Direct Numerical Control mode. This runs programs that are larger than 256K from either the hard drive or a floppy disk without loading it into the buffer. When DNC mode is selected, the screen in Figure 9-5 will appear.

If "*RUN FROM DISK*" is selected, the load menu (Figure 9-3) will appear so that a PGM or TXT file can be selected from the hard drive or floppy disk.

Once a file is selected, it will be loaded into the DNC area of memory and become the active program to run or view.

NOTE: A DNC program may not contain any subprograms or macros. They must be in a separate file and loaded into memory separately.

If "*LOAD DNC MACROS*" is selected, the load menu will appear so that a PGM or TXT file, containing the macros used by a DNC program, can be selected from the hard drive or floppy disc. After the DNC subprograms have been loaded, another load screen will appear prompting for the name of a file to DNC.

NOTE: If ESC is pressed at any time during the DNC prompting process, DNC mode will be aborted. Also, the letters "DNC" will be displayed next to the program number when the DNC mode has been successfully entered.

*** SET Z = 0** This command resets the Z axis to zero. This is used to determine where the top of the part is. Before executing a program, bring the tool down so that it touches the top of the part, then press the * key to reset the Z axis. This

should also be done each time the tool is changed during the execution of the program. Doing this will also set the TLO of the active tool equal to the distance the tool is from the QUILL UP position. (ie. if the tool is in the QUILL UP position, the TLO will equal zero.)

[+] SET This command will save the option settings and return to the RUN mode screen.

[ESC] EXIT This will exit the SET OPTS screen without saving any of the setting changes and return to the RUN mode screen.

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

/SET Z=0 This key command is used to set the Z display to zero **in 2-axis mode only**. Doing this will also set the TLO of the active tool equal to the distance the tool is from the QUILL UP position. **This command does not appear in 3-axis mode.**

/Q UP This key is used to move the quill up, away from the part, to its top-most position. **This command functions in 3-axis mode only and does not appear in 2-axis mode.**

DLT Display When the **RUN** mode is selected, the EZTRAK logo in the upper left corner of the screen changes to show an X, Y, and Z value. This display has the letters **DLT** in the lower right corner. These values show the Distance Left to Travel for the current instruction, shown in the **ACTV** line in the lower portion of the screen. These values should not be confused with coordinates, nor programmed X, Y, and Z values.

REAL-TIME GRAPHICS

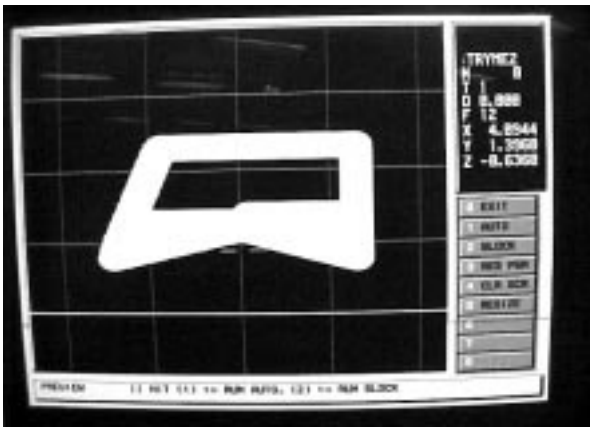


Figure 9-6

If Preview is selected while the machine is running a part program, the software enters into Real Time Graphics mode. The"pen" draws as the table moves, simulating the tool's movement as it executes the cutting operation. The path drawn appears without simulating tool diameter.

Preview functions can cause motion. Pressing [2 BLOCK] causes the machine to process one block of information. If the block causes motion, the pen moves with the machine. Pressing [1 AUTO] causes the machine to execute the part program until HOLD or [2 BLOCK] is pressed. [7 EDIT] is not allowed in real-time mode. The pen also becomes the size of the tool as it simulates the tool path.

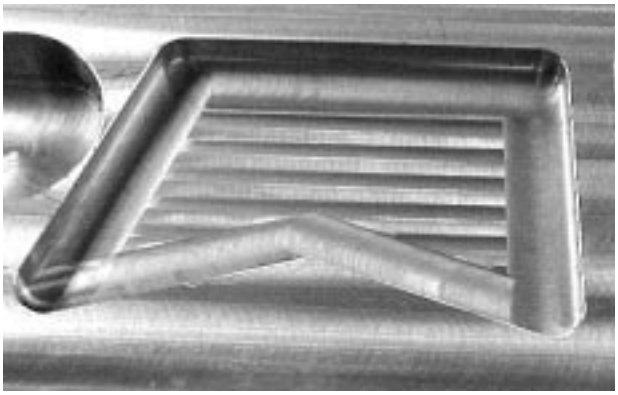
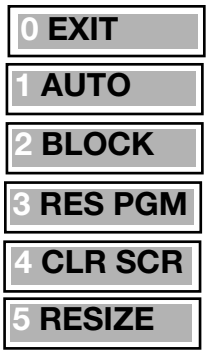


Figure 9-7



Chapter 10
FILE UTILITIES

With File Utility software, the EZTRAK provides a way of managing program files on Disk On Module (or hard drive), floppy drive, remote computer terminals, or other machines via cables & modems. The File Utility commands allow you to update, copy, delete, view, and transfer program files (*.PGM, *.TXT or *.DXF). The commands are accessed by pressing the **9** key [9 UTILS] from the Basic Operations Screen.

The following sections describe how to use the File Utilities Main Window & the individual file utility commands **([0 UPDATE], [1 COPY], [2 DELETE], [3 VIEW], [4 COMM]).**

10.1 FILE UTILITIES MAIN WINDOW [9 UTILS]

Figure 10-1 illustrates the main window of File Utilities which is always displayed over the right hand side of the Basic Operation Screen.

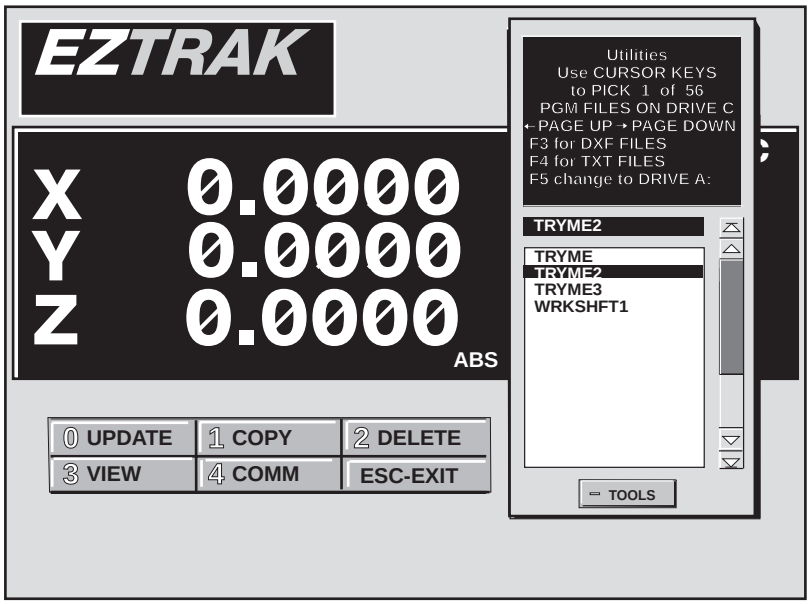


Figure 10-1 File Utilities Main Window

The File Utilities main window is divided into 3 sections or basic steps as shown in Figure 10-2. The steps are described as follows.

STEP 1: **Use the upper section commands to select the proper disk drive location and file type to be used (i.e., ALL *.PGM/*.TXT files or one (1) file to be copied, deleted, viewed or transferred);**

STEP 2: **Use the middle section to select a specific file.**

STEP 3: **Select the desired softkey command ([0 UPDATE], [1 COPY], [2 DELETE], [3 VIEW], [4 COMM]) from the lower section of the window.**

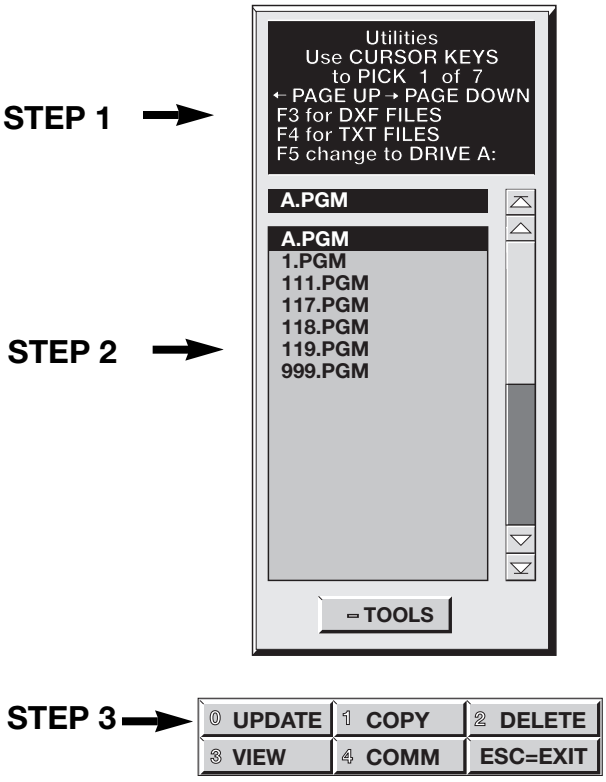


Figure 10-2

Figure 10-2: EZTRAK is listing *.PGM files on the C: drive. This is the default File Utilities [9 UTILS] main window which is obtained by pressing the **9** key [9 UTILS] from the Main Operating Screen.

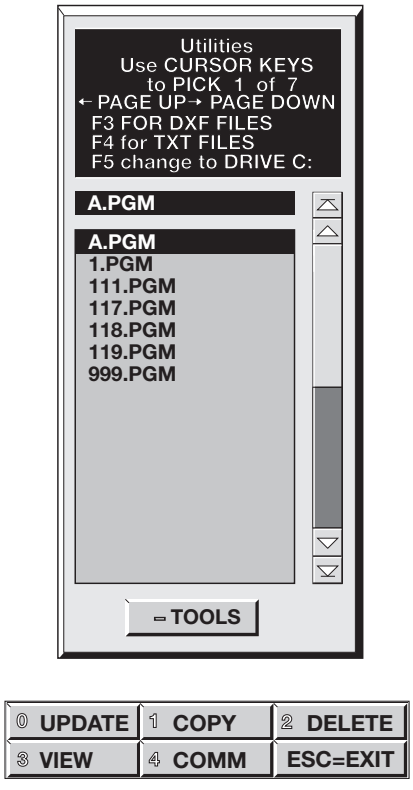


Figure 10-3

Figure 10-3: EZTRAK is listing *.PGM files on the A: drive.

STEP 1:

After entering File Utilities main window, select the proper disk drive location (i.e., A: or C:) and file type (PGM, DXF or TXT). The upper portion of the window is used to do this.

The file types searched for by File Utilities commands are described as follows:

***.PGM** A program file created directly by the EZTRAK machine is identified by having a .PGM extension. This file is written in a "conversational" format and can be created & edited by the **[5 EDIT]** command;

***.TXT** The .TXT extension is a standard text file written in Bridgeport's GCODE (or equivalent) programming format. On the old type control panels, TXT *cannot* be edited using the **[5 EDIT]** command (can be viewed only).

***.DXF** A DXF file is a drawing which the EZTRAK software translates as a sequence of numbers or points connected by interpolated lines. into a PGM file.

The EZTRAK always remembers the last loaded file and the file selection screen will come up with the last loaded file highlighted. If that file was deleted or located on another floppy disk the EZTRAK defaults to listing .PGM files on the C: drive, as shown in Figure 10-2 The EZTRAK will always revert back to this screen when you enter File Utilities from the Main Screen. (The actual files shown in Figure 10-2 will vary depending upon what is actually loaded on your machine.)

If this is not where your files are located, the upper portion of the window can be changed by selecting the **F3, F4**, **F5, or** [–] keys. **F3, F4** and **F5** keys are "*toggle*" keys. *Toggle* keys are used to switch between two options; the **F4** key switches between .TXT and .PGM file types, and the **F5** key switches between the A: and C: drives. The **F3** key toggles between DXF and PGM files. The **[–]**key switches to .TBL file types (Tool Library files).

For example, from the Figure 10-2 window press the **F5** key to change to the A: drive. EZTRAK will then search the A: drive for all *.PGM files. You will briefly see a yellow message box "**SCANNING PATH...**" in the upper left hand side of the screen while the EZTRAK is searching. If there are *.PGM files on the A: drive, then the File Utilities window will become as shown in Figure 10-3.

If there are no *.PGM files on the disk in the A: drive (or if there is no disk in the drive), you will see a quick message box "**NO FILES FOUND A:**" or **"DRIVE NOT READY: READING A"** and you will default back to listing the PGM files on Drive C.

If you press the **F4** key, the EZTRAK will change from searching for *.PGM files on the A: drive to *.TXT files as shown in Figure 10-4.

NOTE: Keep in mind that the EZTRAK is always reading the opposite of what is listed in the upper portion of the File Utilities window for the F3, F4 and F5 keys.

The EZTRAK automatically scans for the presence of Drives A:, C: & D:. Therefore, if your EZTRAK machine is equipped with a hard drive in addition to a disk on module, then you will see an additional command in the upper portion of the File Utilities main window for the **F6** key -"**F6 for changing to DRIVE D:".**

Likewise, in the copy **[1 COPY]** & delete **[2 DELETE]** command windows, you will have the additional option to read Drive D: by pressing the **F6** key.

To exit the File Utilities main window, press the **ESC** key.

Once the desired file location and type are selected, then the next step (**STEP 2**) is to use the cursor keys to search the list and select (i.e., highlight) the desired file. **STEP 2** is described in the following Section, Cursor Movement Keys.

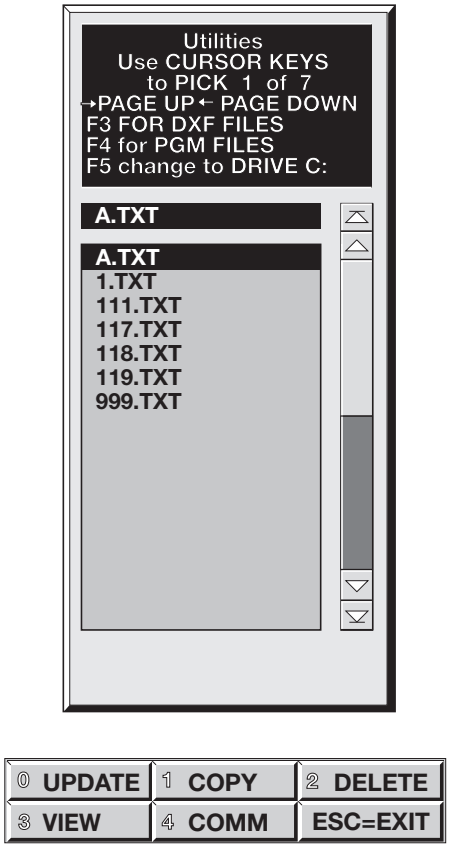


Figure 10-4: EZTRAK is listing *.TXT files on the A: drive.

STEP 2:

10.1.1 Cursor Movement Keys

When the desired list of files is displayed in the File Utilities window, then you can use the cursor (arrow) keys to select an individual file.

Use the Page Down [↓] and Page Up [↑] cursor keys to scroll through the list of files. Use the Previous [←] and Next [→] cursor keys to page up and page down through the list of files by pages.

NOTE:: Remember the arrow keys on the keyboard are labeled ['] & ['] but will actually move the cursor up and down by LINES, not by pages.

To copy or delete ALL *.PGM or *.TXT files: Use the **F3** key command shown in the upper portion of the **1** and **2** command windows.

STEP 3:

The last basic step (**STEP 3**) to using the File Utilities window is to select the desired softkey command. These commands allow you to Update Software **[0 UPDATE]**, Copy **[1 COPY]**, Delete **[2 DELETE]**, View **[3 VIEW]**, Communicate (i.e., transfer) **[4 COMM]** files; and Exit **[ESC=EXIT]** the File Utilities main window.

The following sections describe these commands which can be used to manage your program files.

10.2 UPDATE FILES (SOFTWARE) [0 UPDATE]

Figure 10-5 shows the dialog box which is displayed when the **0** key **[0 UPDATE]** is pressed from the File Utility main window. This command is **ONLY** used when you want to update your machine with a new version of software. All updates are loaded from the A: drive. If there is no disk in the A: drive, then an message will be displayed: "****ERROR** Floppy not found. Hit any key to continue"**, and you will return to the Main Operating Screen.

Follow instructions, notes and warnings as shown when loading software updates.

Figure 10-5

10.3 COPY FILES [1 COPY]

NOTE: The 1 command window is a 3-step procedure like the 9 File Utility main window.

The EZTRAK File Utilities **[1 COPY]** command allows you to copy a single file or ALL *.TXT/ *.DXF/*.PGM files from either the C: or A: drives. In addition, a complete copy of a back-up disk can be made using the diskcopy command **[2 DISK COPY]**.

The EZTRAK scans your hardware setup and only gives you options which are available. For example, most machines will only have a C: drive and an A: drive so that when you have files listed from the C: drive in the File Utilities window, you will only have the option to copy to the A: drive.

NOTE: If both a flash disk and a hard drive are present, then you will see options for a DRIVE D: The disk on module will always be the C: drive location.

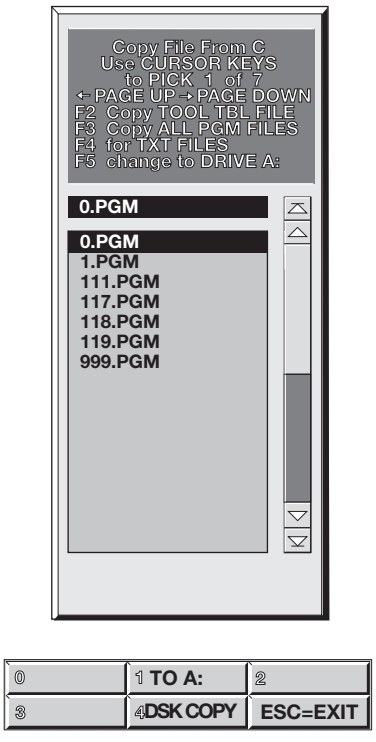


Figure 10-6

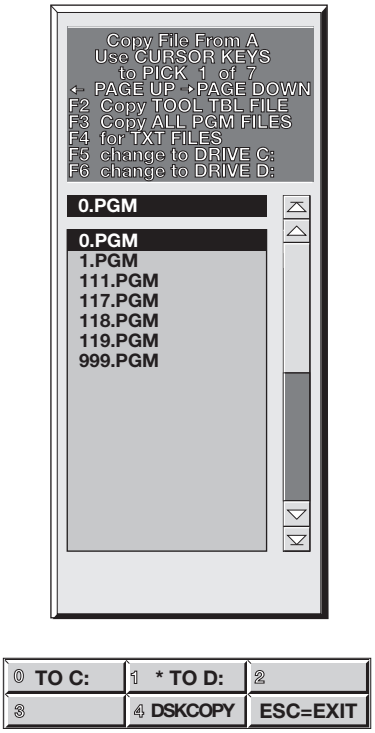


Figure 10-7

As with most EZTRAK commands, the operator will be prompted to confirm or cancel the operation by pressing the **ESC** key **[ESC=EXIT].**

The direction of file transfer (i.e., from C: to A: or from A: to C:) is changed in the upper portion of the copy command window.

The following is a typical example of how the copy **[1 COPY]** procedure is performed for copying a file from the C: drive to a floppy A: drive.

1.. Select a file (i.e., 0.PGM) to be copied from the list (Figure 10-6) using the ↑, ↓, ←, and → cursor keys so that it is highlighted.

2. Press the **[1]** key **[1 TO A:]** and Figure 10-8 will be displayed.

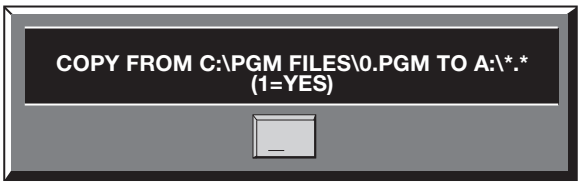


Figure 10-8

3. Type 1 and press the **ENTER** key. The following message box will be displayed in the upper left hand corner : **"WAIT... Copy in Progress"** Figure 10-6 will again be displayed. Press the **ESC** key **[ESC=EXIT]** once to exit back to File Utilities, then press **ESC** again to exit back to the Main Operating Screen.

NOTE: The copy process will only proceed if the number one (1) is typed in the prompt box of Figure 10-8. EZTRAK will return you to the Figure 10-6 window if anything other than 1 is entered.

The Copy ALL *.PGM (or ALL*.TXT) files command (i.e., **F3** key) works in the same way as described above for copying single files only the * (i.e., wildcard) character is used to represent all files. For example, *.TXT or *.PGM represents all files with a .TXT or .PGM extension; *.* represents all files with any extension type.

10.3.1 DISKCOPY [2 DISK COPY]

Program files and related data are usually stored on the machines' disk on module (or hard disk). It is recommended to periodically back-up these onto floppy disks using the copy command **[1 COPY]** as discussed in the previous section. However, it is also highly recommended that you prepare a second back-up of this disk.

Use the diskcopy command **[4 DISK COPY]** to copy an entire floppy disk (i.e., 3.5 inch) onto another one. This can be used to create a back-up disk for floppies containing program files or the EZTRAK system or Software Update disks.

> **NOTE: If you are copying the EZTRAK system disk, make sure the system disk is "locked" before copying. Locking the disk will prevent it from being accidentally overwritten. To LOCK a disk, slide the small tab on the back of the disk so that the hole is uncovered. If you need to re-start the EZTRAK, remember to slide the tab back so the hole is covered up.**

The diskcopy process will require swapping the disks as you would using the DOS command version of diskcopy to create a backup disk of your program file disk or EZTRAK system disk. Remember that the ***Target*** disk is the ***blank*** disk and the ***Source*** disk is the disk containing the information to be copied.

For example, to make a copy of a disk in Drive A:

1. From the 9 UTILS Main Window, press the **1** key for the Copy command.
2. Press the **4** key for the diskcopy command **[4 DISK COPY]** and your

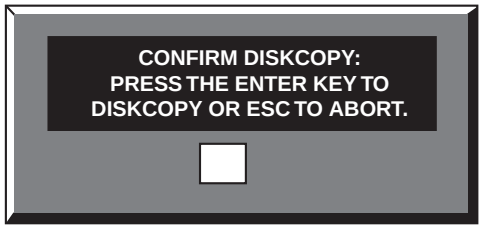


Figure 10-8a

screen will ask you to confirm the command:

3. Press Enter to continue and your screen will display the following message.

Insert SOURCE diskette in drive A:
Press any key to continue...

NOTE: Before proceeding, make sure a disk is in Drive A and that it is not write-protected or it will be necessary to reboot your machine.

4. Place the system disk or diskette to be copied in the drive. If the disk cannot be read or is missing, then you will see the following message.

   Not ready - A:, Make sure a diskette is inserted into the drive and the door is closed. Press CTRL + C to abort or correct this problem and press any key to continue...

   Unfortunately, there is no C key or CNTL key on the EZTRAK keyboard, so only desktop versions can be returned to the **[9 UTILS]** Main Window by typing **CTRL-C. Without a full keyboard, you must reset the machine if the above message appears.**

5. After the source disk is properly placed in the A: drive, the following message will be displayed:

   Copying 80 tracks, 18 sectors per track, 2 side(s)
   Reading from source diskette....

6. After a minute, the following message will be displayed prompting you to insert a Target disk, that is, the blank disk to be copied to.

   Insert TARGET diskette in Drive A:
   Press any key to continue...

7. After pressing a key you will see this message:

   Writing to target diskette...

8. You will then be asked if you want to make another copy of the same diskette.

   Do you wish to write another duplicate of this disk (Y/N) ?

   If so, you will be returned to Step 6 above.

   If not, you will see a message documenting the Volume Serial Number for the disk, and you will be asked if you want to copy another disk.

   Volume Serial Number is ####-####
   Copy another diskette (Y/N) ?

   If you do want to make a copy of a different disk, type **Y** and you will be returned to Step 2 above.

   If not, type **N** and you will be returned to the Main Menu.

IMPORTANT: There is no way to abort the diskcopy [D] command once it has begun on the EZTRAK. The best an operator can do is allow the machine to follow through even if you use the same disk as source and

target. Desktop versions can use the CTRL-C command to abort at any time because they have a full size keyboard.

10.4 DELETE FILES [2 DELETE]

NOTE: The [2 DELETE] command window is a 3-step procedure like the [9 UTILS] File Utility main window.

Figure 10-9 illustrates the default **[2 DELETE]** window.

The EZTRAK File Utilities **[2 DELETE]** command allows you to delete a single or All *.TXT/ *.PGM files from either the C: or A: drives. It works similar to the **[1 COPY]** command with the following exceptions:

The **1** key command **[1TO A:]**, **[1TO C:]** (i.e., drive location) is replaced by the **0 SELECT** command.

Pressing the **0** key **[0 SELECT]** will display Figure 10-10 when the default window is open. When the "**F3 Delete ALL TXT (PGM) FILES**" command is selected, then by pressing the **0** key **[0 SELECT]** Figure 10-11 will be as shown.

These windows give the operator an opportunity to

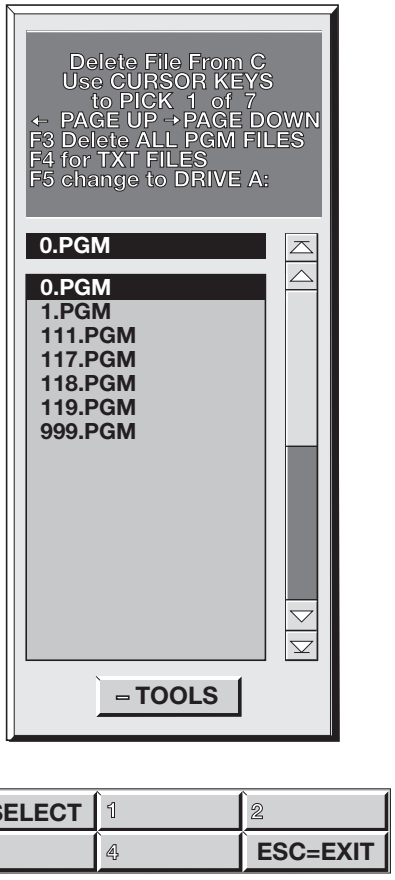


Figure 10-9. The default 2 window

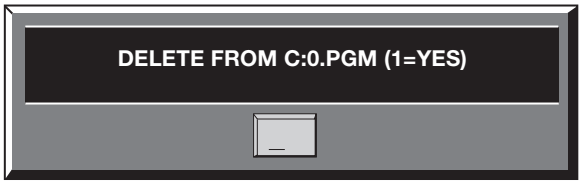


Figure 10-10. This window will confirm the deletion of a single file named 0.PGM from the C: drive. Press the 1 key to delete the file, typing anything else will cancel the deletion.

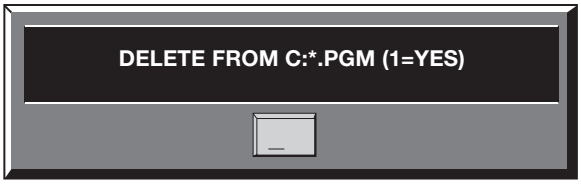


Figure 10-11. This window will confirm the deletion of ALL files with a .PGM extension from the C: drive. Again, Press the 1 key to delete all .PGM files, typing anything else will cancel the deletion.

cancel the deletion if necessary by pressing the **ESC** key **[ESC=EXIT]** or pressing any key ***EXCEPT*** the **1** key.

10.5 VIEW FILES [3 VIEW]

The EZTRAK File Utilities **3** command allows you to view the selected (i.e.,highlighted) *.PGM,*.TXT or *.DXF file from the File Utilities main window list. This command will only allow you to VIEW the text of the selected file. Press the **5** key [EDIT] from the Main Operating Screen if you want to actually modify a *.PGM file. (Recall that *.TXT files *cannot* be edited).

10.6 COMMUNICATIONS [4 COMM]

The EZTRAK is capable of transferring files to and from another machine or a PC using RS-232 port connections. Methods for copying files are accessed from the [9 Utilities] screen and the [4 COMM] key. When 4 (Communications) is selected, a screen comes up with a choice of 2 options:EZLINK or ASCII.

< 1 > EZLINK is used for communicating with another Bridgeport Machines product (ie., EZTRAK, EZCAM & EZFILE). Bridgeport Machines EZ-UTILITIES 6.2.1 is the most commonly used utilities software when communicating through EZLINK and EZCAM to a remote PC device. It is available from your local Bridgeport Machines Dealer and is described below in detail.

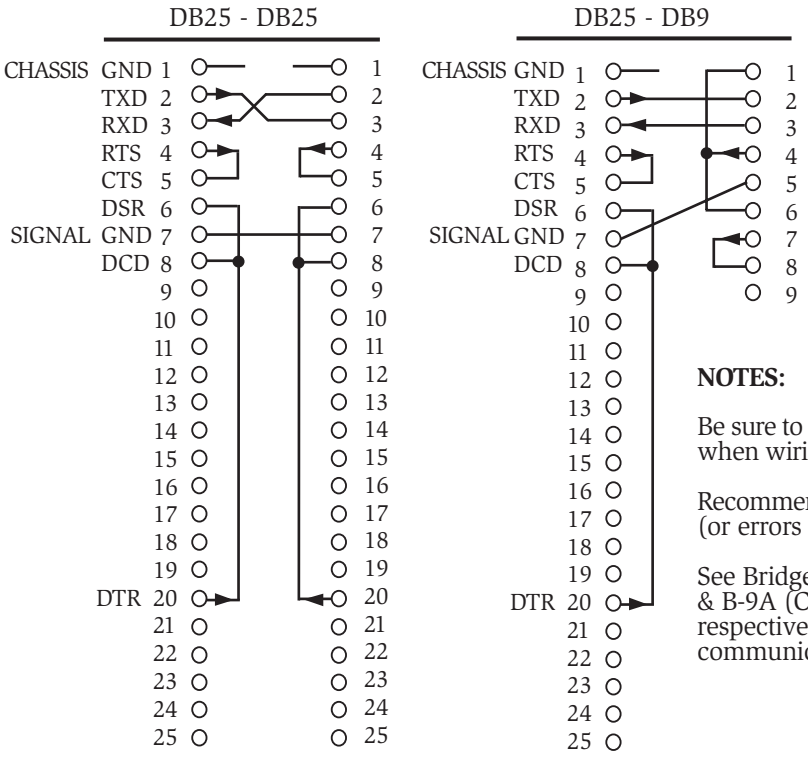
< 2 > ASCII is used to communicate with a remote PC device other than a Bridgeport Machines product. Communications software is available with the CAM System or can be found in your Windows version (ie., Terminal in Windows 3.1 and Hyper Terminal in Windows ___ and above.)

There are 2 basic steps which need to be taken prior to using the [4 Comm] command to transfer files for the first time.

STEP 1: Make sure you have the proper cable connections between the EZTRAK and the other device (PC or Machine).

The most common, direct and easiest way to transmit files is to use Bridgeport Machines' Universal Communications cable (Part No. 1940303). This cable can be used alone when communicating from one EZTRAK to another. The cable

COMMUNICATION CABLE PIN ASSIGNMENTS



NOTES:

Be sure to correctly match pin & socket connections when wiring the cables for your particular devices.

Recommended total cable length: Less than 50 feet (or errors may occur during transmission).

See Bridgeport Machines Accessory Bulletins B-9 & B-9A (Code. No.'s 1104-1349 & 1104-1440, respectively) for further information regarding communication cables.

SERIAL PORT PINOUTS: The necessary communications cables for connecting the computer to any device or CNC control can be made with standard parts available from most electronic supply stores. The pin functions for a standard 25-pin RS-232 port and a standard 9-pin RS-232 port are shown below. Check you computer owners' manual for the correct information.

RS-232 25-PIN SERIAL PORT ASSIGNMENT

1. Frame Ground — Protective Ground
2. Transmit Data — (TXD)
3. Receive Data — (RXD)
4. Request To Send — (RTS)
5. Clear to Send — (CTS)
6. Data Set Ready — (DSR)
7. Signal Ground — (GND)
8. Data Carrier Detect — (DCD)
9. Positive Voltage
10. Negative Voltage
11. Select Standby
12. Secondary Received Line Signal Detector
13. Secondary Clear to Send
14. Secondary Transmit Data
15. Transmit Signal Element Timing
16. Secondary Receive Data
17. Receive Signal Element Timing
18. Test
19. Secondary Request to Send
20. Data Terminal Ready — (DTR)
21. Signal Quality Detector
22. Ring Indicator — (RI)
23. Data Signal Rate Selector
24. Data Transmitter Signal Element Timing
25.

RS-232 9-PIN SERIAL PORT ASSIGNMENT

1. Data Carrier Detect — (DCD)
2. Receive Data — (RXD)
3. Transmit Data — (TXD)
4. Data Terminal Ready — (DTR)
5. Signal Ground — (GND)
6. Data Set Ready — (DSR)
7. Request to Send — (RTS)
8. Clear to Send — (CTS)
9. Ring Indicator — (RI)

Figure 10-12

connects to the RS232 serial ports on each machine to complete transfer communications.

When communicating between an EZTRAK and a remote PC device, the EZ-CAM adapter cable (Part No. 1940515) can be used. It is connected to the Universal Communications cable and plugged in at the remote PC device. These cables are optional parts which can be obtained from your Sales & Service Center or Bridgeport Machines Dealer. A description of Communication Cable Pin Assignments are described below.

If you are not sure whether or not you are "connected", check to see if the cables are attached to the serial port of the machine and the communicating device (PC or another EZTRAK).

The three most common communication cable pin connections used today are: DB25-DB25; DB25-DB9; and DB9-DB9. Figure 10-12 illustrates how the EZTRAK utilizes these typical kinds of cable connections. Please refer to this information and your computer owners' manual if you need to obtain cable parts from a local electronics supply store.

NOTE: The EZTRAK does not check for proper connections before trying to send a file. Therefore, if you do try to use [4 COMM] file utilities and are not properly connected you will most likely get an error message such as: "ERROR CANNOT OPEN COMM PORT" when you try to SEND or RECEIVE a file.

TROUBLESHOOTING NOTE : If you are sure that your cable connections are correct but you are still not able to transmit files, FIRST check to see that the connections are secure, that is - are not loosely fitted together. Occasional movement to the machine may loosen connections so always check this first if the machine has been able to transmit files in the past.

Use a standard RS-232 signal tester (25 pin) or VoltOHM meter to verify that the proper pin assignments are indeed receiving a signal. For the signal tester, the appropriate lights will be on when the connections match the pin assignments. A VoltOHM meter can be used to verify that signals (i.e., voltage) are being transmitted.

If you are still not able to transmit files, then check that your communication settings are correct and that you have all the necessary software programs properly loaded. These procedures are described in STEP 2 .

STEP 2: Set general software communication parameters (Baud Rate, Serial Port, Data; Parity etc.) and transfer protocols the same for both the EZTRAK Machine and the other device.

Each machine needs to use the same protocol so that the two machines can communicate properly with each other. Bridgeport Machines Inc. products generally support many different protocols (EZ-CAM, Heidenhain, BOSS 5, Fanuc, etc.). However, for simplicity, the EZTRAK is primarily designed to use EZ-LINK to communicate with another EZTRAK machine, or EZ-FILE / EZ-CAM software on a PC. ASCII protocol can also be used.

The EZTRAK machine software is designed to configure itself (i.e., set communication parameters) for transferring files, and will display these settings just prior to conducting the transfer. It is up to the operator to assure that the other sending/receiving device is configured the same.

The following sections summarize general communication settings for each communication protocol as required by the EZTRAK.

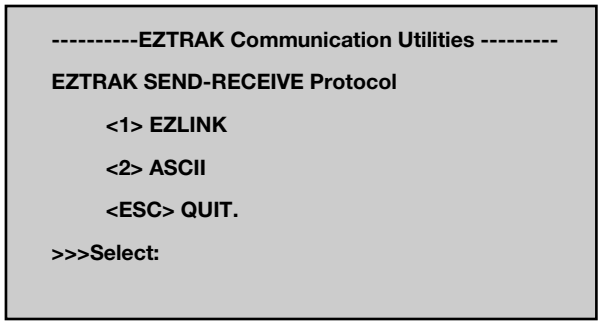


Figure 10-13 The first of the 2 main communication screens displayed when the **4** command is used. This screen is used to specify which transfer protocol will be used. (If ASCII is selected, the window in Figure 10-16 will appear next.)

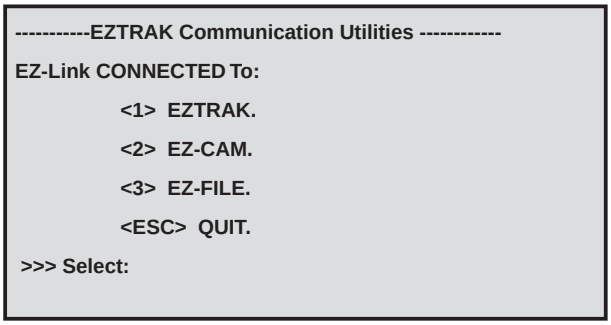


Figure 10-14 When <1> EZLINK is selected in Figure 10-13, this screen appears. The operator must specify what device the EZTRAK machine will be connected to.

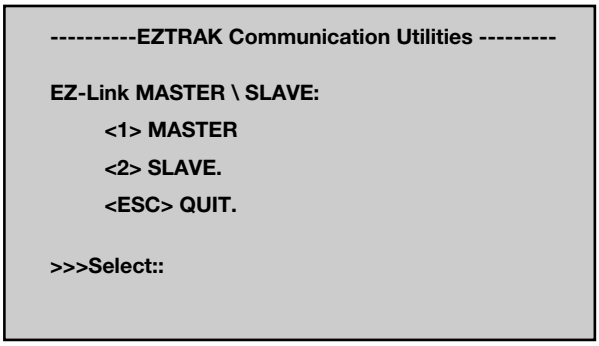


Figure 10-15 If <1> EZTRAK is selected in Figure 10-14, this window appears. If you are using EZ-LINK to transfer files to another EZTRAK (or Bridgeport Machine product), you will need to specify whether your machine is the “master” device or the “slave” device.

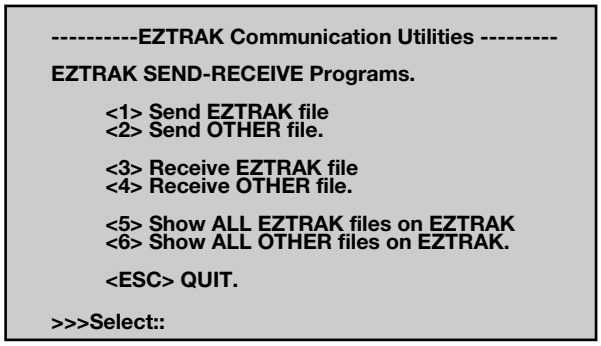


Figure 10-16 Each of the transfer protocols will use this screen. The operator is asked to determine whether files will be sent or received by the EZTRAK.

10.6.1 Transferring Files Between EZTRAK Machines using EZ-UTILITIES Version 6.2.1

Sending files from one EZTRAK to another, using EZLINK and EZTRAK:

At the receiving machine:

Select 9 UTILS from the Basic screen.

Select 4 Comm

< 1 > EZLINK.

< 1 > EZTRAK

< 2 > Slave

< 3 > Receive EZTRAK File. (Conversational pgm file), or

< 4 > Receive OTHER File.(G-Code txt file)

At “Enter EZTRAK File name to receive” or “Enter OTHER File name to receive” type the name of the file and press the Enter key.

The screen will say “Wait - - Receiving File: C:\PGMFILES\ (file name)”.

At the sending machine:

Select 9 UTILS from the basic screen.

Select 4 Comm

< 1 > EZLINK

< 1 > EZTRAK

< 1 > Master

At “Enter EZTRAK File name to send” or “Enter OTHER File name to send”, type the name of the file and press Enter. The screen will say “Wait - - Sending File (name).

The file will scroll across the EZTRAK screen as it is being received, and will say “Transfer Complete” when it is finished.

10.6.2 Sending a PGM File from the EZTRAK to a Remote PC Device using EZLINK and EZ-CAM

NOTE: All files on the PC will be received with a txt extension regardless of whether it is a pgm file (Conversational) or a txt file (G-Code). When transferring a file to the EZ-TRAK, it is the operator’s responsibility to select the correct directory:

“Send/Receive EZTRAK File” refers to a **pgm** file;
“Send/Receive Other File” refers to a **txt** file.

When sending a file from the EZTRAK to a computer,set up the computer first. When sending a file from the computer to the EZTRAK, set up the EZTRAK first.

At the Ez-Utils 6.2.1 screen, select Settings from the Menu Bar and then Serial Port.

Set Baud Rate to 4800

Set Port to Com 1

Press OK

Select “To CNC” from the Menu Bar

Select EZ-LINK and a window will come up saying “Ready For Data Transfer!”

At the EZ-TRAK screen Select:

< 1 > EZ-LINK

< 1 > EZ-TRAK

< 1 > MASTER

< 1 > SEND EZ-TRAK FILE

Type the name of the file and press Enter.

The file will scroll across the EZ-TRAK screen as it is sent and a message will say “Transfer of File Complete” when transmission is complete.

10.6.3 Sending a txt file from the EZTRAK to a remote PC device using EZ-LINK and EZ-CAM

Follow the same steps as for sending a pgm file except select < 2 > Send OTHER File.

10.6.4 Receiving a pgm file at the EZ-TRAK from a remote PC device using EZ-LINK and EZ-CAM.

At the Ez-Utils 6.2.1 screen, select Settings from the Menu Bar and then Serial Port.

Set Baud Rate to 4800

Set Port to Com 1

Press OK

Select "To CNC" from the Menu Bar

Select EZ-LINK and a window will come up saying "Ready For Data Transfer!"

At the EZ-TRAK screen select:

9 Utilities

4 Comm

< 1 > EZ-LINK

< 2 > EZ-CAM

< 3 > Receive EZ-TRAK File.

At the prompt type the name of the EZ-TRAK file to be received at the machine and press enter. (The name at the receiving end does not have to be the same as at the transmitting end.)

The EZTRAK screen will say "Wait - - Receiving File: C:\ (followed by the program name)".

At the remote device the directory window will come up. Highlight the file to send or type the name at the prompt and press the Open key. A window at the remote device will come up saying "Data Transfer in Progress!".

The file will scroll across the EZTRAK screen as it is being received, and will say "Transfer Complete when it is finished.

10.6.5 Receiving a txt file at the EZTRAK from a Remote PC Device using EZLINK and EZ-CAM

Follow the same steps as for sending a pgm file, except select < 4 > Receive OTHER File.

10.6.6 Transferring files to and from the EZTRAK using EZ-LINK and EZFILE

The use of a "Greco Box" is required for transferring files via EZLINK and EZFILE. There fore it is necessary to refer to your Greco Box owners manual for more information.

10.6.7 Using a remote PC device to transfer files to and from an EZTRAK via ASCII

1. Sending a pgm file (Conversational) or txt file (G-Code) to a remote PC device using ASCII protocol.

Set the remote PC device settings as follows:

Baud Rate = 9600
Data Bits = 8
Stop Bits = 2
Parity = None
Flow Control = X on/X off
Port = Com 1 or Com 2 accordingly.
Set the communications software to receive the file.

At the EZTRAK machine:

Select 9 Utilities
Select 4 Comm
< 3 > ASCII
Select one of two choices: < 1 > Send EZTRAK File, < 2 > Send OTHER File.
Enter the file name to send.
The EZTRAK Communications screen will say " Transfer of (name of file) Complete" when it has finished.

2. Receiving a pgm file (Conversational) or txt file (G-Code) to the EZTRAK using ASCII protocol.

Set the remote PC device settings as follows:

1.) Baud Rate = 9600
2.) Data Bits = 8
3.) Stop Bits = 2
4.) Parity = None
5.) Flow Control = X on/ X off
6.) Port = Com 1 or Com 2 accordingly.

At the EZTRAK machine:

Select 9 utilities

Select 4 Comm

< 3 > ASCII

Select one of two choices: < 1 > Receive EZTRAK File or

< 2 > Receive Other File.

Enter the File name to receive and press Enter.

The EZTRAK Communications screen will say " Transfer of (name of file) Complete" when it has finished.

10.7 COMMON MESSAGES

This section is a listing of most of the most common messages (including error messages) that can appear at different times on the EZTRAK screen. These messages typically appear within small yellow boxes in the upper left had side of the screen.

A description of when the message would occur and what the message means is provided.

COM ERROR: COMM PORT 1 NOT SET

This message is displayed when there is a loose cable connection to port 1 or when the connection has not been established.

> > COMM PORT1 9600,8,NO,1 || OPEN

This message appears when a successful link has been established. MODEMs with baud rates up to 9600 bits per second can be used.

Communication Error
Remote Cancel

This message is displayed when the communications link between EZTRAK and a remote terminal was cancelled by the remote terminal.

COPY disk in FLOPPY DRIVE

This message appears when the operator elects to do a complete disk copy. This message lets him know that a complete disk copy is in progress.

DONE RECEIVING FILE PGM-file-name

Notifies the operator the receive operation has completed successfully.

DONE SENDING FILE file-name

This message notifies the operator that the send operation has completed successfully.

Drive not ready

A floppy disk could not be found in drive A:\. Check that there is a floppy disk in the disk drive, that the disk is pressed in, and that the drive door is closed all the way.

END OF PROGRAM FAULT: EZ-LINK communications aborted

This message is displayed when a link could not be established between the EZTRAK and a remote terminal.

ERROR: BREAK TRAP NOT SET

This message is displayed when " < 7 > SEND or RECEIVE files" has been selected, followed by " < 3 > ASCII" or " < 2 > YMODEM" and a link could not be established between the EZTRAK and a remote terminal. Specifically **i**nterrupt **s**ignal **h**andling (ISH) could not be set. ISH allows a send or receive operation to be aborted by pressing the **escape** key.

ERROR: Cannot Save Received File

The file that was received from a remote machine could not be saved on drive C:\. Make sure that drive C:\ can be written to.

ERROR: DESIGNATED FILE [PGM file-name] NOT FOUND
ERROR: DESIGNATED FILE [TXT file-name] NOT FOUND

The file to be sent could not be found on the EZTRAK.

ERROR IN SENDING FILE

An error was encountered while attempting to send a file to a remote device.

ERROR RECEIVING FILE PGM-file-name

An error was encountered while attempting to receive a file from a remote device.

File Read Error

This message appears in **[9 UTILS]** if an I/O error occurs while sending or receiving a file. Check the disk sectors being used for the file being sent or received on the sending machine if you are sending a file and on the remote machine if you are receiving a file.

File Write Error

The operator receives this message during a send or receive operation in **[9 UTILS]**. EZTRAK could not write the data for the file being sent to the destination machine or could not write the data for the file being received to the receiving machine.

HIT < + > TO DISKCOPY

This prompts the operator for confirmation and gives him a chance to bail out of the disk copy operation.

NOTE: COPY will involve swapping the original disk and the blank disk 3 times.

This message informs the operator of what the disk copy operation involves.

NO FILES FOUND *.PGM
NO FILES FOUND **.TXT

This message appears if, after the root directory on the A:\ or C:\ drive has been searched for PGM or TXT files, no files were found having the "PGM" or "TXT" extensions.

Read fault

An error was detected while reading from a floppy or hard drive. Run a diagnostic test on the floppy or hard drive.

SCANNING PATH... *.PGM
SCANNING PATH... **.TXT

This message appears immediately after the operator selects **[5 EDIT]** from the Main screen or **I** from the RUN Mode. The file directory on the C:\ drive or the root directory on the A:\ drive (depending on whether "F5 change to drive A:" has been selected) is searched for all file names having the "PGM" or "TXT" extension (depending on whether "F4 MORE FILES" has been selected). After the search is completed, the operator is presented with a list of files to choose from. This message informs the operator to wait until the file search on the A:\ or C:\ drive has completed.

User Cancel

The operator receives this message after **[9 UTILS]** has been selected and a communications link has been established. It is displayed when the operator uses the **escape** key during a send or receive operation.

WAIT –– SENDING protocol-type FILE [file-name] –– HIT [ESC] TO ABORT

Where protocol type can be:

1. EZ-LINK – the communication format used by Bridgeport equipment.
2. YMODEM – a type of tele-communication protocol.
3. ASCII – sends an ASCII file straight through without any modifications.

A send operation is in progress. To abort the operation, press the **escape** key.

WARNING||
WRITE PROTECT the original disk before copying.

This message appears as a safety precaution to protect the original disk before the copy operation takes place.

Write fault

An error was detected while writing to a floppy or hard drive. Run a diagnostic test on the floppy, hard drive or flash disk.

Chapter 11
TEACH MODE

INTRODUCTION

The **TEACH** mode of the EZTRAK is a simple way of saving a part program as it is performed manually. The operator cuts the part manually, using the EZTRAK in the JOG mode or the POWER OFF MANUAL mode, and saves a point at the end of each milling move or drilling event. The EZTRAK stores each point and can save the entire program on the diskette for later recall and editing. The points (0–99) are saved as they are created in a file called TEACH.PGM.

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

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

NOTE: Teach mode can only be accessed from the Basic Operation screen with POWER OFF. A message "Set Power Off to Save Points" appears on the screen until axis power is shut off.

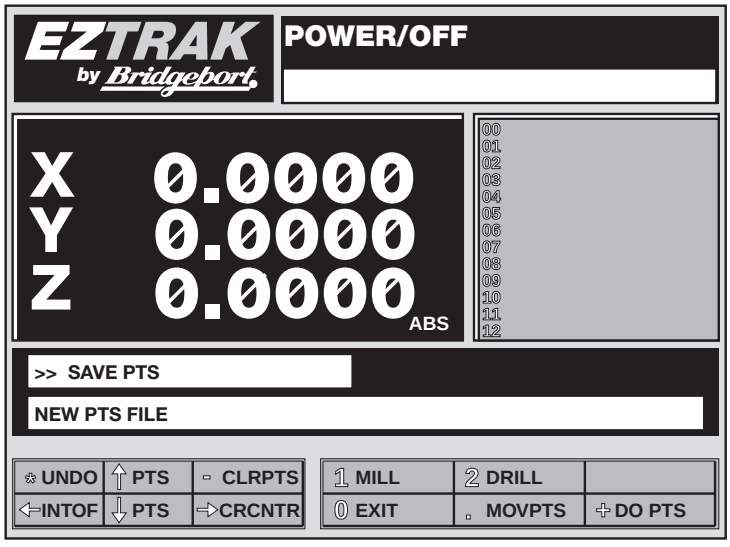


Figure 11-1

The commands available from the TEACH screen are:

1 MILL

Press the **1** key to save the current point as a **MILL EVENT**. This means that the EZTRAK will mill a straight line to this point from the previous location. Arcs cannot be saved as mill events.

2 DRILL

Press the **2** key to save the current position as a **DRILL EVENT**. The EZTRAK will make a positioning move to this point when the points are replayed, and then stop and wait so that the hole can be drilled at this location (in 2-axis mode) or drill the hole automatically (in 3-axis mode).

+ DO POINTS

This key creates a **PGM** from the saved points in the same order that they were created. Immediately after **DO POINTS** is selected, a data box appears that allows the user to enter the DO PTS parameters. These parameters are: **Z UP** (the Z clearance for positioning moves), **Z MILL** (the Z depth for milling events), **Z DRILL DEPTH** (the depth of each hole), **F FEED** (the speed at which the operation is performed for milling events and the speed at which the tool moves into the part for drilling events). After the data in the parameter box is complete, **TEACH.PGM** is created and loaded into memory so it can be executed. The TEACH mode is then exited and the RUN mode becomes active so the program can be executed.

. MOVPTS

The **.** key allows moving the tool to a point that was previously saved. The system prompts the user to select one point in the points list. After the point is selected, hitting the **.** key and **ENTER** moves the tool to that point.

– CLR POINTS

The – key deletes all of the saved points from the points list.

* UNDO

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

0 EXIT

The **EXIT** key exits the **TEACH** mode and returns to the BASIC OPERATIONS screen. When the TEACH mode is exited, the programmed points are only saved in the points list and not in **TEACH.PGM**. To create TEACH.PGM, DO PTS must be selected.

NOTE: If EZTRAK, is shut down or UTILS is entered, the information in the points list will be lost.

<←> INTOF

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

<→> CRCNTR

This command finds the center of a circle that is defined with three consecutive points in the points list. This point can then be stored in the points list as a separate point. When this command is selected, the user is prompted for the number of the first of the three points in the points list. The user is then shown the coordinates of the new point, and the system prompts for the type of point (mill or drill) and the number where this point is to be stored.

<↑> PTS

This command scrolls the points list upward. The top of the points list is the 00 point.

<↓> PTS

This command scrolls the points list downward. The bottom of the points list is the point 99.

If the operator enters the **TEACH MODE** from the **JOG** mode, the **3** key will have the function of **JOG Y**. If the **TEACH MODE** is entered from the **BASIC OPERATIONS** screen, the **3** key will have no function.

Chapter 12
PREVIEW MODE

INTRODUCTION

The **PREVIEW** mode is used to pre-view the cutter path of a part program. The control panel screen shows an XY or XYZ view of the part and displays tool movements so that the program can be checked before it is executed.

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

12.1 THE PREVIEW SCREEN

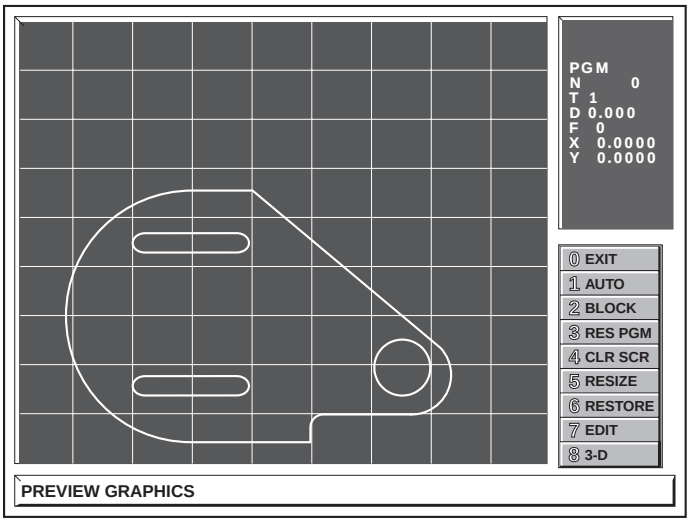


Figure 12-1

12.2 KEY FUNCTIONS

0 EXIT

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

1 AUTO

The **AUTO** command sets the loaded program to be viewed in continuous operation. The preview halts for programmed stops, or tool changes, but does not stop for Z axis adjustment. The program displays the tool path when the **AUTO** key is pressed.

2 BLOCK

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

3 RES PGM

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

4 CLR SCR

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

5 RESIZE

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

Xmin	the left edge of the part
Xmax	the right edge of the part
Ymin	the back edge of the part
Ymax	the front edge of the part
Zmin	the bottom edge of the part
Zmax	the top edge of the part

NOTE:
The Zmin and Zmax values are only used when the 3D view mode is engaged.

The **Xmin**, **Ymin** and **Zmin** values can be set to the lower left back corner of the part. The **Xmax**, **Ymax** and **Zmax** values can be set to the upper right front corner of the part.

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

6 RESTORE

The **RESTORE** command returns the PREVIEW screen to the original size set by the EZTRAK®, after the screen size has been changed using the **RESIZE** command.

7 EDIT

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

8 3D/2D

The **3D/2D** key toggles from two dimensional view to three dimensional view.

USING PREVIEW

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

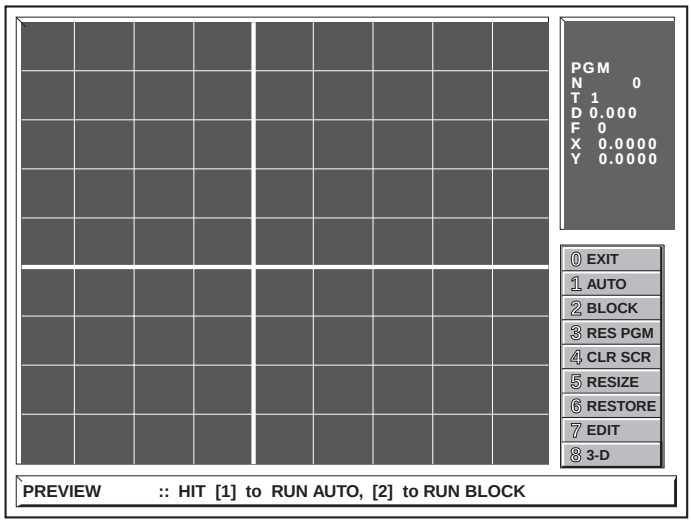


Figure 12-2

Press the **1 AUTO** key to begin simulation of the part program on the screen. The + key must be pressed after programmed stop events and tools changes, just as if the program were actually executing on the machine. Drilling holes requires the + key to be pressed only while the program is actually running, it is not required in PREVIEW.

Milling moves are shown on the screen as solid lines. Rapid moves are shown as dotted lines. Drilling locations are shown as circles with a + symbol at the center.

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

At the beginning of a program, the operator is prompted:

|| HIT [1] to RUN AUTO, [2] to RUN BLOCK

When a tool change is executed the operator is prompted:

|| HIT [+] to CONTINUE, or [0] to STOP

At the end of a program the operator is prompted:

|| HIT [3] to RESET PROGRAM

NOTE:

If a key is pressed while the program is running, “operation aborted” is displayed and the selected function is executed.

Appendix A
Axes and Coordinates

Some Background Information

When we talk about a machine tool cutting a part, we say that the cutter moves in three axes. These are the three directions in which the tool moves in order to cut the material of the part. These directions are left and right (the X axis), forward and backward (the Y axis), and up and down (the Z axis). If you watch the machine tool in motion though, it is the worktable, and the part itself that move (forward and backward, and left and right) not the cutting tool, or cutter.

It is easier to think of the tool moving, so that the motion makes more sense. This way the tool does all of the movement. It also makes sense when you think about the directions and the values they represent on the screen. When the X coordinate is increasing in value, the motion of the tool is to the operator's right. When the Y coordinate is increasing in value, the tool is moving towards the back of the machine.

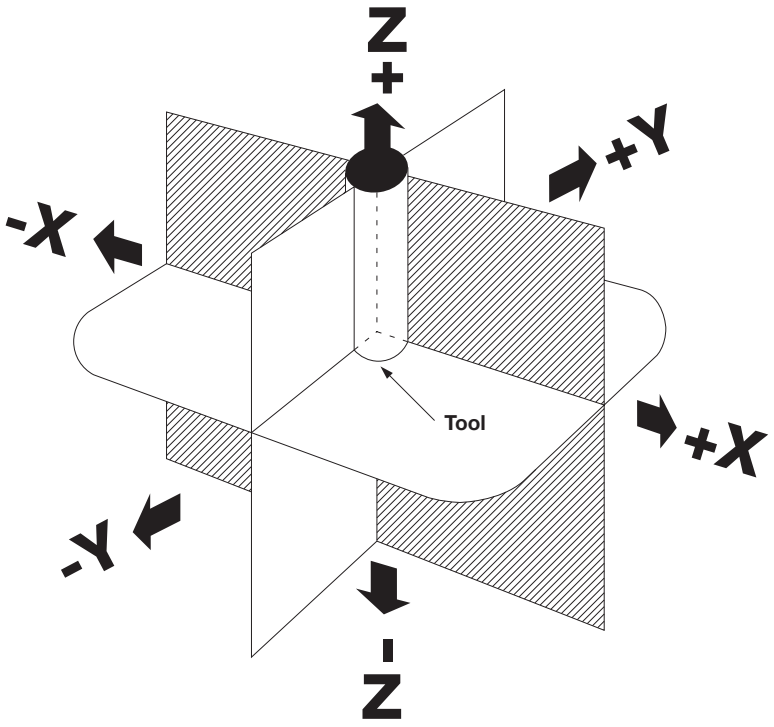


Figure A-1

Axes and Coordinates

We've already mentioned the terms **axis**, and **coordinate**, but what do they have to do with the EZTRAK®?

The EZTRAK® (and all CNC machine tools) use a **coordinate system** to identify where the tool is at any one moment, and to define where the tool is moving to when it is cutting chips. This coordinate system is called a **Rectangular Coordinate System**.

A Rectangular Coordinate System is based on a grid of lines which have set distances from one central point called the **origin**. Every point on the grid is given both an X and a Y coordinate which show how far to move to get to that point from the origin. Coordinates are usually shown in parentheses **( )** with a comma separating them. They are always given in the same order, X first, and then Y. A Z coordinate is added when the point can move along a third axis, called the Z axis. The Z coordinate on the EZTRAK® DX determines how deep the tool is cutting into the part. The Z coordinate is always listed last when the coordinates are shown in parentheses.

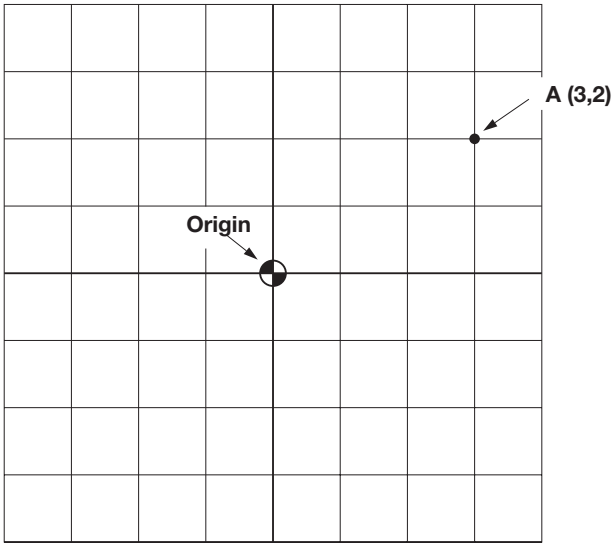


Figure A-2

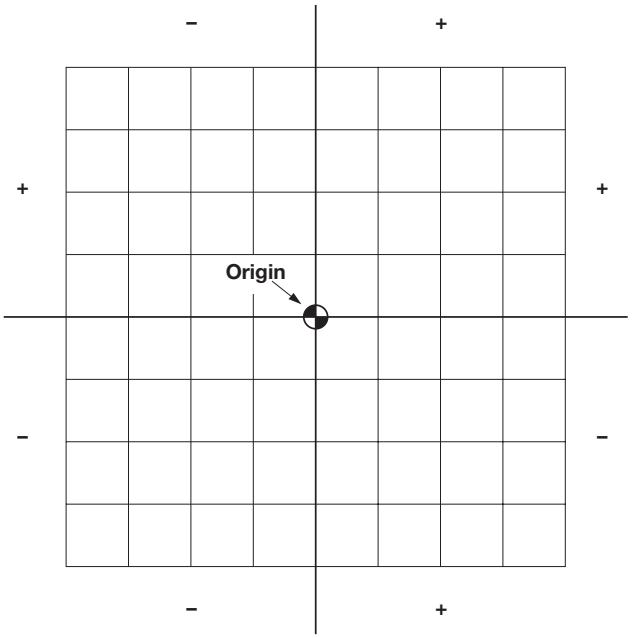


Figure A-3

The origin is given the coordinates (0,0) to make things simple. This means that to move to a point we'll call **A** that has the coordinates (3,2), you would start at the origin and move 3 grid lines (usually inches) along the X axis, and then 2 grid lines along the Y axis. Or you could move 2 inches along the Y axis and then 3 inches along the X axis. You still end up at point **A**, but the path you took to get there is different.

When you move from one point to another on the grid, you count up or down depending on the direction you're moving to get to the desired point. If you move to the right, you count up, or add to the X coordinate. If you move left, you count down, or subtract one for each grid line you cross as you move to get to the target point. If you move towards the top of the grid you add to the Y coordinate, and if you move towards the bottom of the grid, you subtract one for each line that you cross.

If you cross to the left side of the origin, or below the origin, you reach zero and then go beyond it. These are negative numbers and are shown with a minus sign, like -1. The minus in -1 shows that you are one line to the left or below the origin, depending on whether the -1 is the X coordinate or the Y coordinate.

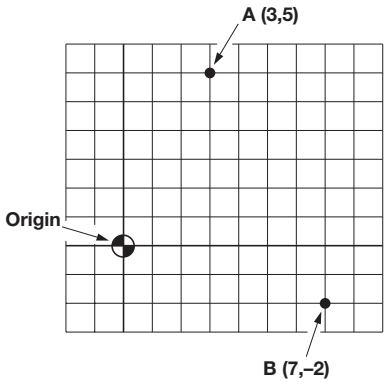


Figure A-4

As you move on the grid the number of lines that you cross in each direction is the distance that you moved along each axis to get to the new point. This gives two distances for each move, one along the X axis, and one along the Y axis. It is easy to calculate the distances necessary to move from one point to another.

Start with the X coordinate of each point. Take the higher number of the two X coordinates and subtract the lower number. Remember that subtracting a negative number is the same as adding a positive number (e.g. 3 – -2 = 5). The answer is the distance between the X coordinates of the two numbers. Repeat this for the Y coordinates. For example, if we take two points (7,-2) and (3, 5) (shown in **Figure A-4**). Take the higher of the two X coordinates and subtract the lower number (7 – 3 = 4). Repeat for the Y coordinates (5 – -2 = 7). The distances are 4 in the X and 7 in the Y.

This doesn't tell you how to move to get to the new point though. If you were at the point (7,-2) and wanted to move to the point (3,5) you would need to know more than just how far to move. You need to know **what direction** to move.

Compare the X coordinates again. You are starting at a point whose X coordinate is 7, and you want to go to a point whose X coordinate is 3. 7 is greater than 3 so the movement direction is towards the lower numbers on the left. This gives the direction along the X axis. The same method is used to find the Y direction. You are starting at a point whose Y coordinate is -2. You are moving to a point whose Y coordinate is 5. Since -2 is less than 5, the movement direction is towards the greater numbers at the top of the grid. This gives the direction along the Y axis.

Absolute vs. Incremental Programming

There are two ways to program a machine tool to move from one point to another. You can instruct the machine to move from the current location to the point whose coordinates are (X_1,Y_1). This is called **absolute** programming because the coordinates you give the machine are based on a known origin (0,0) location.

The second way to program this same move, is to tell the machine tool how far to move in each axis. This means that the point is given as two distances from the current location, one along the X-axis, the other along the Y-axis. Each distance is accepted as a direction and distance. For instance, giving **-2** as the X distance moves the tool to the left where giving **2** would move the tool to the right. Each of these would move the tool the same distance from the current location, but in opposite directions. This is called **incremental** programming, and it does not matter where the origin is located.

In programming the EZTRAK®, you may use either **absolute** or **incremental** programming. Sometimes it is easier to use one or the other. The program mode is shown in the programmed instructions for the EZTRAK® as **ABS** or **INC**. This is usually the third item seen in the instruction line. For example:

```
0000 LINE INC X4.000 Y0.000 Z0.000 F10.
```

This instruction tells the EZTRAK® to mill a line from the current position, four inches in the positive X direction, zero inches in the Y direction, and with the Z set at zero, at a feed rate of 10 inches per minute.

Below is a grid with some points marked on it. The origin is at the center of the grid. Starting at the first point given, move to the next point, and write down the absolute and incremental values of the move. Remember when you move to the right or towards the top of the grid, your movement is positive, when you move left or down, it is negative.

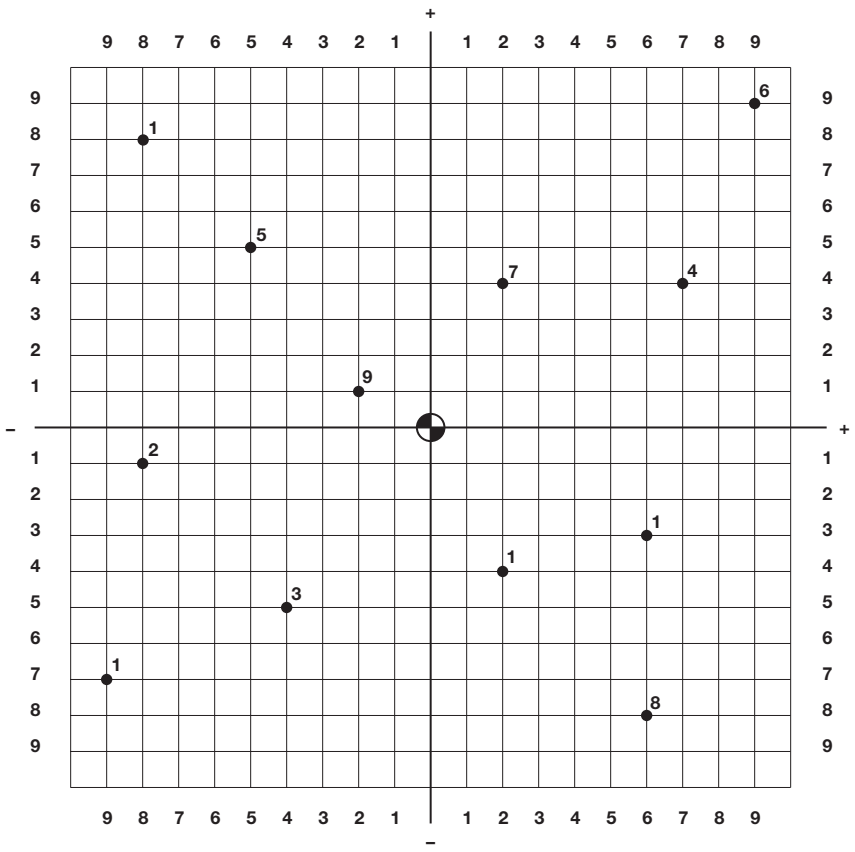


Figure A-5

ABSOLUTE

example:

7 to 11 X -9 Y -7

6 to 1	X_____	Y_____
5 to 4	X_____	Y_____
8 to 3	X_____	Y_____
2 to 4	X_____	Y_____
4 to 5	X_____	Y_____
5 to 6	X_____	Y_____
12 to 7	X_____	Y_____
10 to 8	X_____	Y_____
4 to 9	X_____	Y_____

INCREMENTAL

7 to 11 X -11 Y -11

6 to 1	X_____	Y_____
5 to 4	X_____	Y_____
8 to 3	X_____	Y_____
2 to 4	X_____	Y_____
4 to 5	X_____	Y_____
5 to 6	X_____	Y_____
12 to 7	X_____	Y_____
10 to 8	X_____	Y_____
4 to 9	X_____	Y_____

ANSWERS TO EXERCISES

ABSOLUTE

example:

7 to 11	*X -9*	*Y -7*

6 to 1	X −8	Y 8
5 to 4	X 7	Y 4
8 to 3	X −4	Y −5
2 to 4	X 7	Y 4
4 to 5	X -5	Y 5
5 to 6	X 9	Y 9
12 to 7	X 2	Y 4
10 to 8	X 6	Y −8
4 to 9	X −2	Y 1

INCREMENTAL

7 to 11	*X -11*	*Y -11*

6 to 1	X −17	Y -1
5 to 4	X 12	Y -1
8 to 3	X −10	Y 3
2 to 4	X 15	Y 5
4 to 5	X −12	Y 1
5 to 6	X 14	Y 4
12 to 7	X 0	Y 8
10 to 8	X 0	Y −5
4 to 9	X −9	Y −3

Appendix B
EZTRAK Programming Example

As an example, the following part is given with the programmed instructions. The blueprint for the part is shown below.

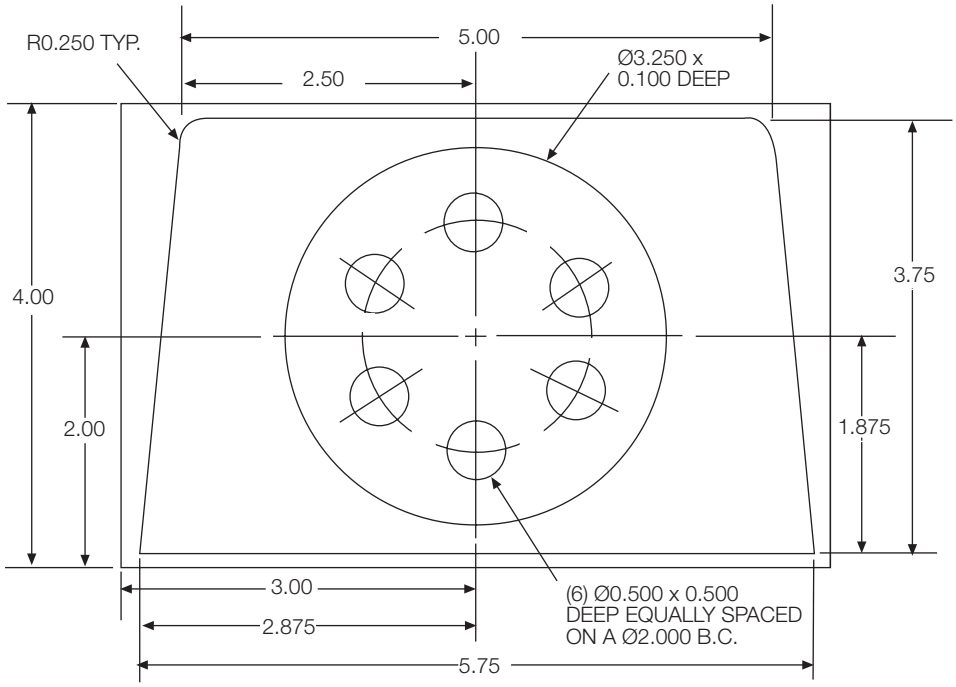


Figure B-1

Select the 8 MDI mode from the BASIC OPERATIONS screen to begin programming the part. This part is cut out of a block of material 6" x 4" x .75". Put a 1/2" end mill into the quill. Make sure Tool 1 is active in the display. Jog the tool down to the top of the part. Set Z = 0 and quill up. Set X = 0 and Y = 0 to the center of the part.

0010 ||TOOLCHG T1

The first line of the program should be a tool change, so that the operator has an opportunity to load the correct tool, for the start of the program. This is essential if the program will be repeated and it requires more than one tool.

0020 COMP|ON LFT D.5 X-2.875 Y-1.875 Z.1 Z-.1 P.5 F10.

This line turns on cutter compensation. The part will be milled clockwise with the cutter on the left side of the part. The starting point will be at the bottom left hand corner of the part. The cutter will rapid to .1 above the part and mill to a depth of .1 from the top of the part. Cutter Comp is engaged with a .5 approach move with a .5 arc from the left side of the start point.

0030 BLEND|LN ABS X-2.5 Y1.875 Z–.1 R.250 CW F10.

This line mills the line and the blend arc connecting the next line.

0040 BLEND|LN ABS X2.5 Y1.875 Z–.1 R.250 CW F10

This line mills the second line and the blend arc connecting the next line.

0050 LINE ABS X2.875 Y-1.875 Z-.1 F10.

This line will mill a striaght line to the bottom right corner of the part.

0060 LINE ABS X-2.875 Y-1.875 Z-.1 F10.

This line will return the tool to the start point.

0070 COMP|OFF Z.1

This line automatically turns off cutter compensation with a rapid move to .1 above the part.

0080 CIRCLE PKT X0. Y0. Z.1 Z.2 Z0. R1.625 P.5 P.02 P.25 D.5 F10. F20.

This block calls a canned cycle to mill a circular pocket 3.250 x .1 deep.

0090 ||TOOLCHG T2

This line prompts the operator to make a change to tool number 2, a drill.

0100 DR|BC R1. XC0. YC0. Z.1 Z.6 Z.2 Z.1 A90. P6. F10.

This block calls a canned cycle to drill a 6 hole bolt circle pattern with a radius of 1.0, a center of X0,Y0, .5 inches deep and the first hole along the Y axis.

0110 ||ENDPRGM

This line resets the part program to the top of the text.

Appendix C
Using the Calculator

CALC

In many of the commands throughout the EZTRAK® software, coordinates are required as entered data. Often it is necessary to calculate the correct coordinates using other numbers, or trigonometric functions. These calculations can be carried out by using the **CALC** mode in the EZTRAK software. This mode is called by pressing the * key on the keyboard whenever coordinates are required.

Figure C-1

The screen displays a calculator-like keypad at the bottom of the screen when the **CALC** mode is called. From this screen, complex algebraic equations, and trigonometric functions can be entered and evaluated. When the equation has been entered, press **Enter** to evaluate the equation. The result is placed in the active coordinate field.

At the top of the **CALC** window is the word **CALC:**. This shows the equation entered. At the right of the window is the numeric keypad, used for entering numbers and simple mathematical functions like addition, subtraction,

division and multiplication using the + , –, /, and * keys.

On the lower left side of the **CALC** window the arrow cursor keys are shown. These are used to enter specific characters. The **< left arrow >** cursor key places an open or left parenthesis into the equation field. Each left parenthesis **(** must be matched by a right parenthesis **)** entered by pressing the **< right arrow >** cursor key.

Complex Functions

When one of the function keys is selected, the screen displays a secondary input field.

[ENTER] FUNC:

Simple arithmetic functions (+,–,*,/) can be input along with the complex functions listed below. Some of the complex functions (**ATAN**, and **DIST**) require two numbers to be entered.

F1 SIN

This command enters the trigonometric **SIN(** function into the equation field. The operand for this function is a number, positive or negative which represents an angle in a right triangle. The angle is defined in degrees (e.g. 37.5).

F2 COS

This command enters the trigonometric **COS(** function into the equation field. The operand for this function is a number, positive or negative which represents an angle in a right triangle. The angle is defined in degrees (e.g. 37.5).

F3 TAN

This command enters the trigonometric **TAN(** function into the equation field. The operand for this function is a number, positive or negative which represents an angle in a right triangle. **Note**: Do not enter **90.** degrees for this function. The TAN of 90 degrees is not defined.

F4 ASIN

This command enters the **ASIN(** function into the equation field. This function returns the arcsin of the entered number. The number entered for this function must be between **-1.0000** and **1.0000**, otherwise an error is returned by the system , and zero is entered for the coordinate field.

F5 DIST

This command enters the **DIST(** function into the equation field. This function requires two numbers. Type the first number, then press **Enter**. Type the second number then press **Enter** again. This function uses the Pythagorean Theorem ($A^2 + B^2 = C^2$) to calculate the length of the third side of a right triangle. The entered numbers are the lengths of the first two sides of the right triangle. The calculated value is the square root of the sum of the squares of the two entered numbers. For example if the entered equation is **DIST(3,4)** then the calculated value is **5.0000**.

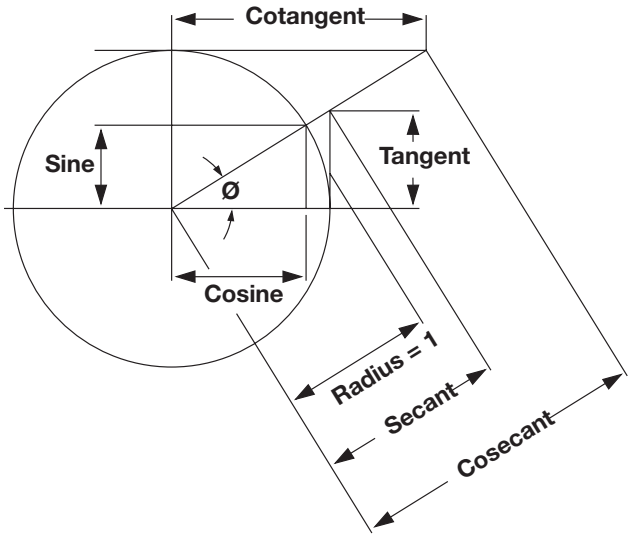


Figure C-2

FORMULAS FOR FINDING FUNCTIONS OF ANGLES
$\frac{\text{Side opposite}}{\text{Hypotenuse}}$ = SINE
$\frac{\text{Side adjacent}}{\text{Hypotenuse}}$ = COSINE
$\frac{\text{Side opposite}}{\text{Side adjacent}}$ = TANGENT
$\frac{\text{Side adjacent}}{\text{Side opposite}}$ = COTANGENT
$\frac{\text{Hypotenuse}}{\text{Side adjacent}}$ = SECANT
$\frac{\text{Hypotenuse}}{\text{Side opposite}}$ = COSECANT
FORMULAS FOR FINDING THE LENGTH OF SIDES FOR RIGHT-ANGLE TRIANGLES WHEN AN ANGLE AND SIDE ARE KNOWN
Length of { Hypotenuse x Sine Hypotenuse ÷ Cosecant Side adjacent x Tangent Side adjacent ÷ Cotangent
Length of { Hypotenuse x Cosine Hypotenuse ÷ Secant Side opposite x Cotangent Side opposite ÷ Tangent
Length { Side opposite x Cosecant Side opposite ÷ Sine Side adjacent x Secant Side adjacent ÷ Cosine

Figure C-3

F6 ATAN

This command enters the trigonometric **ATAN(** function into the equation field. This function requires two numbers separated by a comma, and surrounded by parentheses. The **ATAN** function calculates one angle in a right triangle given the length of the opposite side, and the adjacent side. The first operand should be the length of the side opposite the angle. For example if the entered equation is **ATAN(3,4)** then the calculated value is **36.8966** the measure of the angle opposite the side of length **3**.

<down arrow> SQRT

This command enters the **SQRT(** function into the equation field. This function finds the square root of the entered value. If a negative number is entered, the system returns an error and the equation value is set to zero. The number entered in the coordinate field is zero.

<up arrow> EXP

This function is used to enter an exponent. The exponent must be a whole number and must not be negative. The system returns the value of the operand if the exponent is either negative or contains a decimal. Fractions also may not be used to express an exponent.

Trigonometric Functions

The six trigonometric functions are derived using a circle with a radius of 1, and a right triangle which lies inside the circle. The relationships of the sides and angles can be calculated using various formulae, as shown in the table and illustration below.

Appendix D
GEOMETRY HELP

INTRODUCTION

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

CALLING UP THE GEOMETRY HELP SCREEN

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

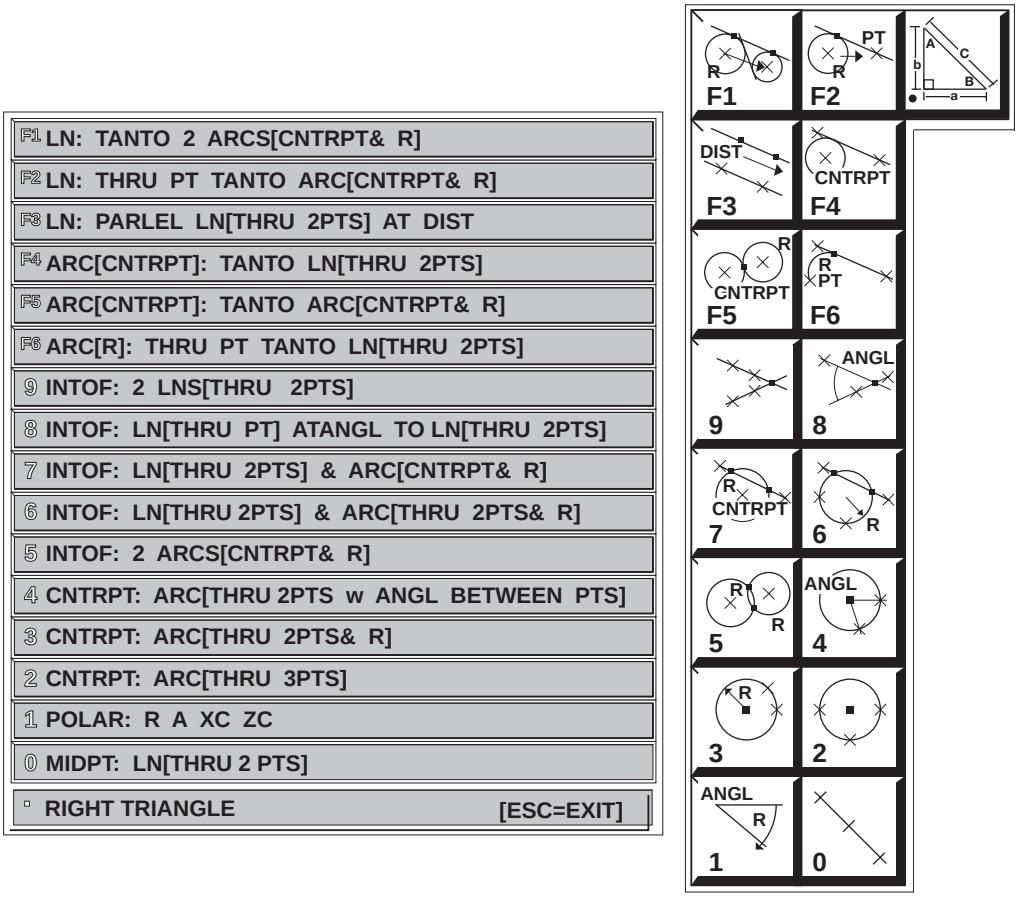


Figure D-1

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

USING GEOMETRY HELP

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

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

GEOMETRY HELP COMMANDS

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

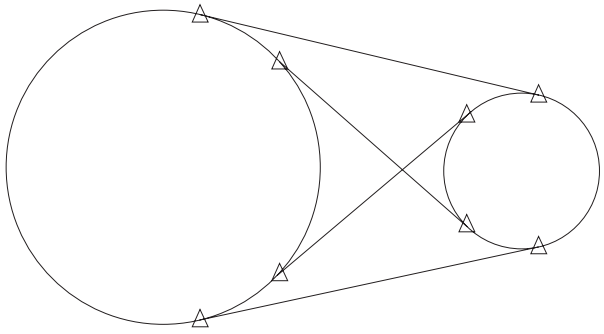


Figure D-2

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

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

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

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

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

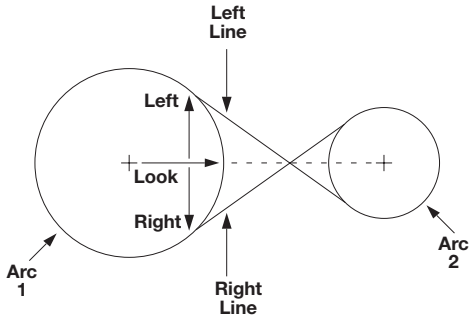


Figure D-3

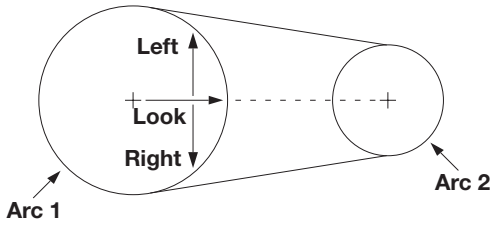


Figure D-4

ARC [1]:

XCntr The X coordinate of the center point of arc 1.

YCntr The Y coordinate of the center point of arc 1.

Radius The radius value of arc 1.

ARC [2]:

XCntr The X coordinate of the center point of arc 2.

YCntr The Y coordinate of the center point of arc 2.

Radius The radius value of arc 2.

Figure D-5

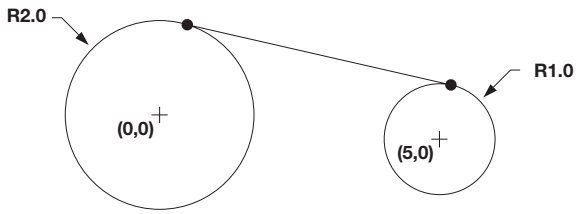


Figure D-6

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

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

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

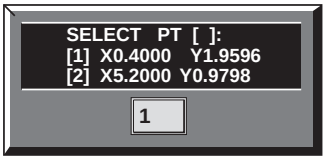


Figure D-7. The Calculated Intersection Points (See Figure D-6)

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

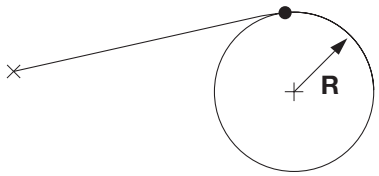


Figure D-8

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

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

The two lines fall on either side of the arc. These lines are specified as being on the left and right of the arc.

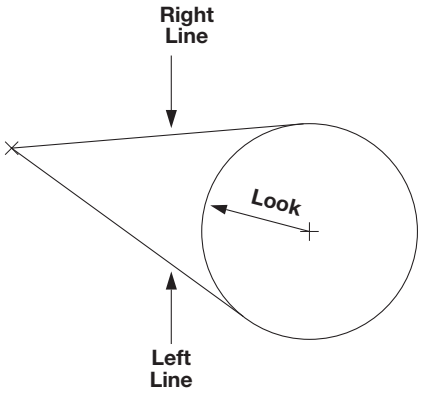


Figure D-9

The direction is determined by looking from the center of the arc towards the specified point, as shown in Figure D-9.

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

In the example shown in the **F2** window (figure D-10) the direction of the line is **left** (See figure D-11). The **Select DIR** parameter has a value of **1** because the line is on the left.

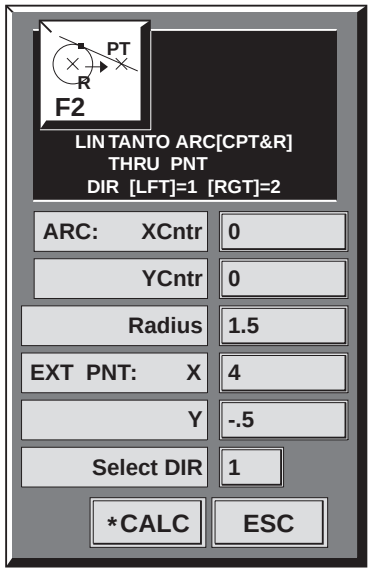


Figure D-10

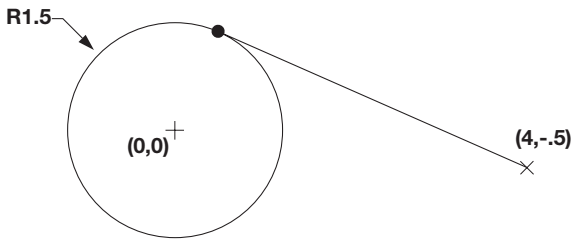


Figure D-11

ARC :
XCntr This is the X coordinate of the center point of the arc.
YCntr This is the Y coordinate of the center point of the arc.
Radius This is the radius value of the arc.

EXT PNT:
X This is the X coordinate of the external point.
Y This is the Y coordinate of the external point.

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

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

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

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

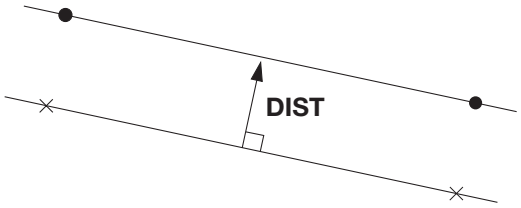


Figure D-12

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

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

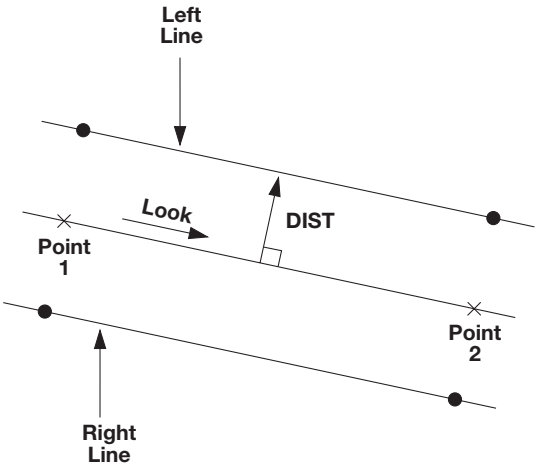


Figure D-13

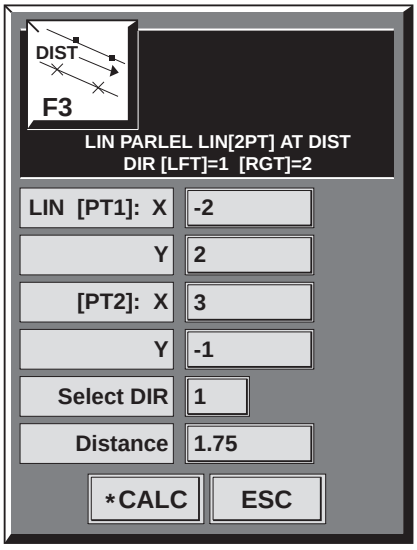


Figure D-14

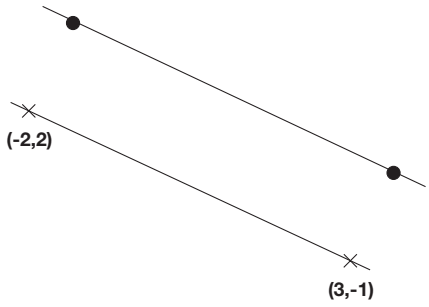


Figure D-15

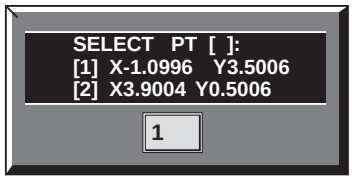


Figure D-16

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

LIN [PT1]:

X This is the X coordinate of the first point of the line.

Y This is the Y coordinate of the first point of the line

[PT2]:

X This is the X coordinate of the second point of the line.

Y This is the Y coordinate of the second point of the line.

Select DIR

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

Distance

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

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

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

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

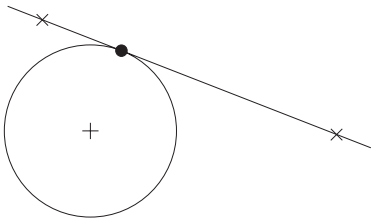


Figure D-17

ARC:

XCntr This is the X coordinate of the arc's centerpoint.

YCntr This is the Y coordinate of the arc's centerpoint.

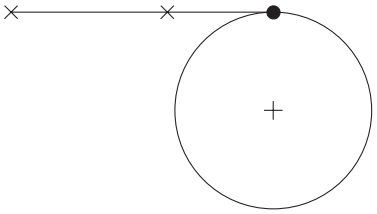


Figure D-18

LIN [PT1]:

X This is the X coordinate of the first point of the line.

Y This is the Y coordinate of the first point of the line.

LIN [PT2]:

X This is the X coordinate of the second point of the line.

Y This is the Y coordinate of the second point of the line.

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

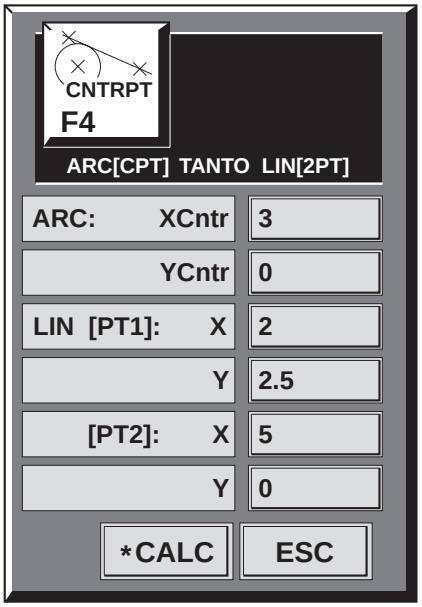


Figure D-19

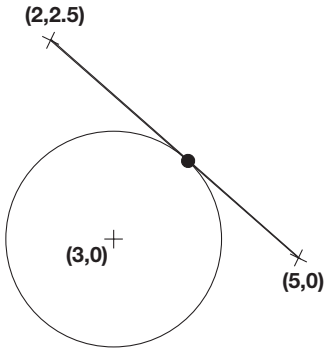


Figure D-20

entered automatically into the correct data fields.

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

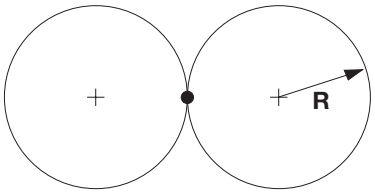


Figure D-21

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

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

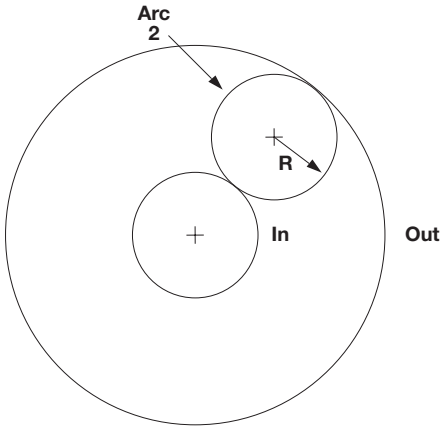


Figure D-22

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

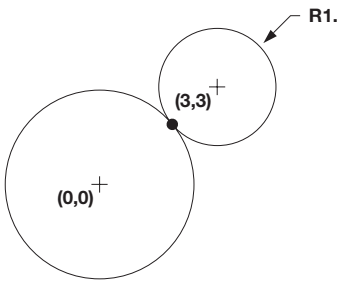


Figure D-23

Figure D-24

In the example shown in Figure D-24, the desired arc is **In**. This arc is selected by entering a **1** in the **Select IN/OUT** parameter, shown in Figure D-23.

ARC[1]:

XCntr This is the X coordinate of the arc's centerpoint.

YCntr This is the Y coordinate of the arc's centerpoint.

ARC[2]:

XCntr This is the X coordinate of the second arc's centerpoint.

YCntr This is the Y coordinate of the second arc's centerpoint.

Radius This is the radius value of the second arc.

Select IN / OUT

This parameter chooses whether the first arc is inside the second arc, or outside.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection of the two arcs, and the radius value of the first arc. The coordinates of the point are entered automatically into the correct data fields of the command.

F6: ARC [R] : THRU PT TANTO LN [THRU 2 PTS]

This command finds the intersection point of an arc, defined by its radius and a point on the arc, and a tangent line, defined by two points on the line. Figure D-25 shows how the arc and the line are defined. The calculated intersection point is shown with a •.

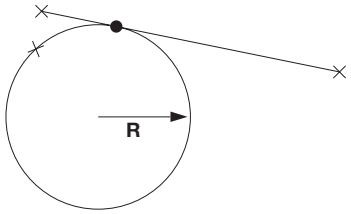


Figure D-25

Note: The arc can be positioned so that its direction may be either clockwise or counter-clockwise. This is shown in Figure D-26. The direction is determined by moving from the defined point towards the tangent point of the line and arc.

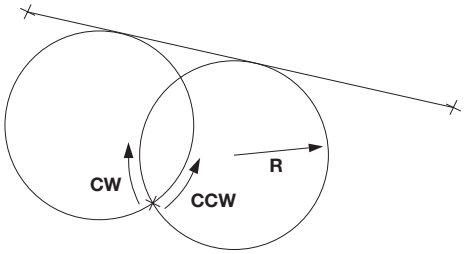


Figure D-26

thru PT:

X This is the X coordinate of the point on the arc.

Y This is the Y coordinate of the point on the arc.

Radius This is the radius value of the arc.

LIN [PT1]:

X This is the X coordinate of the first point on the line.

Y This is the Y coordinate of the first point on the line.

LIN [PT2]:

X This is the X coordinate of the line's second point.

Y This is the Y coordinate of the line's second point.

Figure D-27

Select DIR This parameter chooses the direction of the arc, either clockwise, or counter-clockwise.

When all of the data is entered, and **Enter** is pressed, the system calculates the intersection of the line and the arc (the arc direction is also shown), and the arc centerpoint. These points are displayed as shown in Figure D-29, so that one can be selected as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure D-29. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

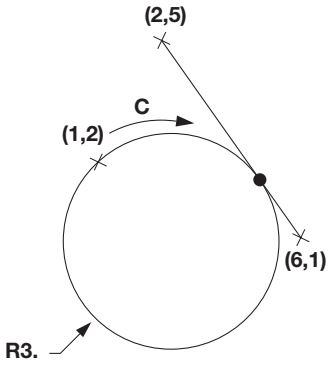


Figure D-28

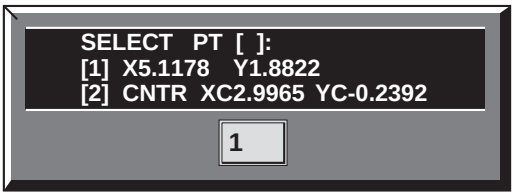


Figure D-29

9: INTOF: 2 LNS [THRU 2 PTS]

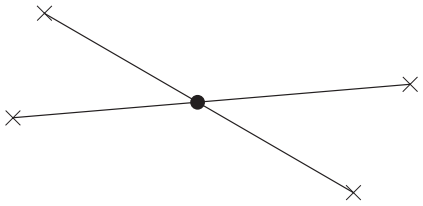


Figure D-30

This command finds the intersection point of two lines. Each of the two lines is defined by two points. This is shown in Figure E-30.

LIN1 [PT1]:

X This is the X coordinate of the first point on line 1.

Y This is the Y coordinate of the first point on line 1.

LIN1 [PT2]:

X This is the X coordinate of the second point on line 1.

Y This is the Y coordinate of the second point on line 1.

LIN2 [PT1]:

X This is the X coordinate of the first point on line 2.

Y This is the Y coordinate of the first point on line 2.

LIN2 [PT2]:

X This is the X coordinate of the second point on line 2.

Y This is the Y coordinate of the second point on line 2.

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

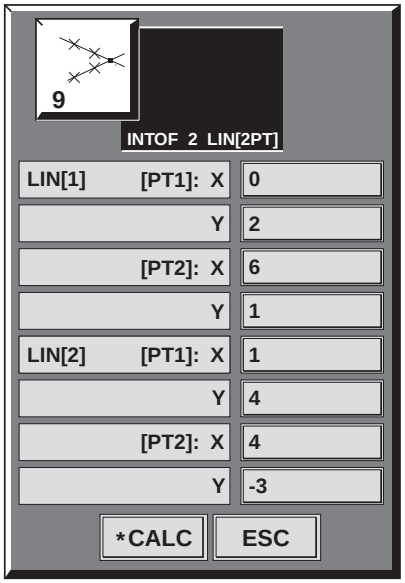


Figure D-31

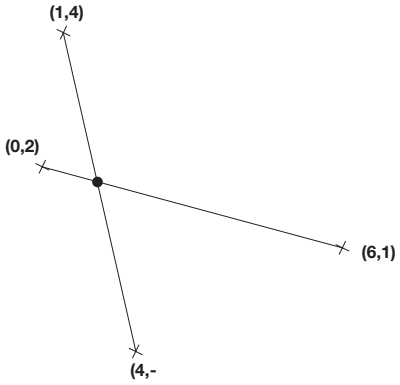


Figure D-32

8: INTOF: LN [THRU PT] ATANGL TO LN [THRU 2 PTS]

This command finds the intersection point of a line, defined by two points on the line, and a second line, defined by a point on the line, and the angle between the two lines. An example of this is shown in Figure D-33.

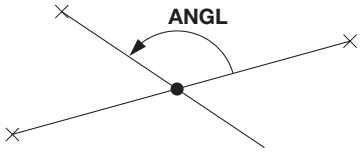


Figure D-33

LIN1 [PT1]:

X This is the X coordinate of the first point on line 1.

Y This is the Y coordinate of the first point on line 1.

[PT2]:

X This is the X coordinate of the second point on line 1.

Y This is the Y coordinate of the second point on line 1.

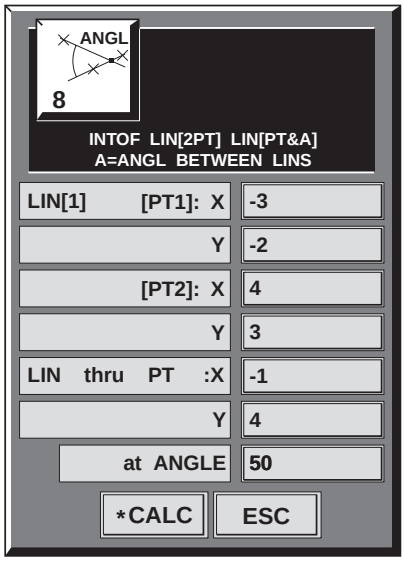


Figure D-34

LIN thru PT:

X This is the X coordinate of the point on line 2.

Y This is the Y coordinate of the point on line 2.

at ANGLE

This is the angle between line 1 and line 2.

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

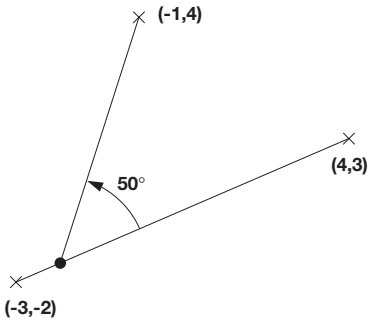


Figure D-35

7: INTOF: LN [THRU 2 PTS] & ARC [CNTRPT & R]

This command finds the intersection points of a line, defined by two points, and an arc, defined by its centerpoint and radius. See Figure D-36.

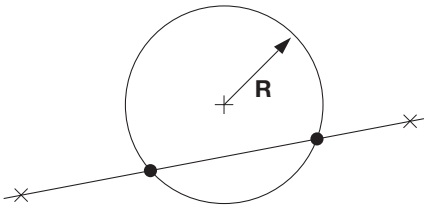


Figure D-36

Note: This command calculates two point locations, giving the user a choice between the two. The two calculated points are shown in Figure D-39.

LIN [PT1]:

X This is the X coordinate of the first point of the line.

Y This is the Y coordinate of the first point of the line.

LIN [PT2]:

X This is the X coordinate of the second point of the line.

Y This is the Y coordinate of the second point of the line.

ARC:

XCntr This is the X coordinate of the arc's centerpoint.

YCntr This is the Y coordinate of the arc's centerpoint.

Radius This is the radius value of the arc.

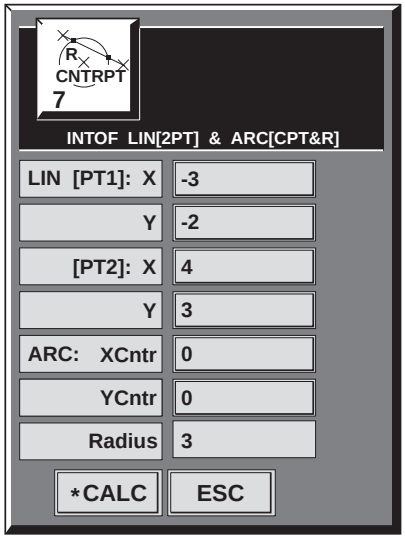


Figure D-37

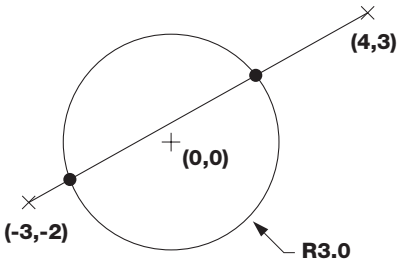


Figure D-38

When all of the data is entered, the system calculates the two intersection points for the line and arc. The coordinates of the two points are displayed as shown in Figure D-39. One of these points can be chosen as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure D-39. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

Figure D-39

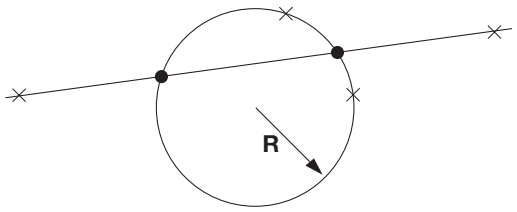


Figure D-40

6: INTOF: LN [THRU 2 PTS] & ARC [THRU 2 PTS & R]

This command calculates the intersection points of a line, defined by 2 points, and an arc, defined by two points on the arc and its radius value. This command returns two intersection points, because the line and arc intersect in more than one location. An example of this is shown in Figure D-40. The user is given a choice of the two points to use as a location in a program or DO EVENT command.

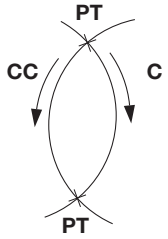


Figure D-41

Note: The direction of the arc must be specified as one of the parameters in this command. The direction is determined by moving around the arc from point 1 to point 2. The direction of this movement determines the direction of the arc. An example of this is shown in Figure D-41.

LIN [PT1]:

X This is the X coordinate of the first point of the line.

Y This is the Y coordinate of the first point of the line.

[PT2]:

X This is the X coordinate of the second point of the line.

Y This is the Y coordinate of the second point of the line.

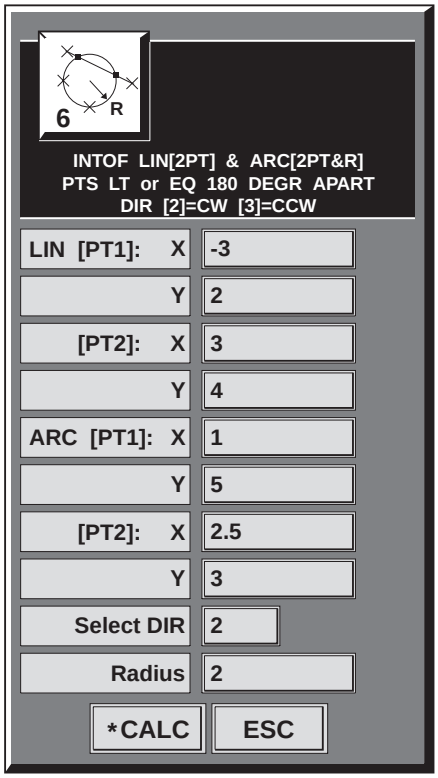


Figure D-42

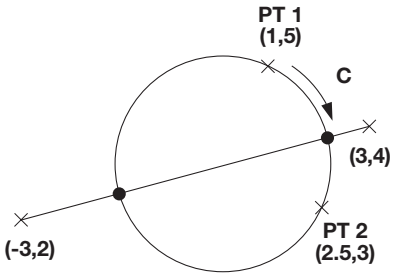


Figure D-43

ARC: [PT1]:

X This is the X coordinate of the first point of the arc.

Y This is the Y coordinate of the first point of the arc.

[PT2]:

X This is the X coordinate of the second point of the arc.

Y This is the Y coordinate of the second point of the arc.

Select DIR

This is the direction of the arc.

Radius

This is the radius of the arc.

When all of the data is entered, the system calculates the two intersection points of the arc and line. The coordinates of the two points are displayed as shown in Figure D-44. One of these points can be chosen as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure D-44. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

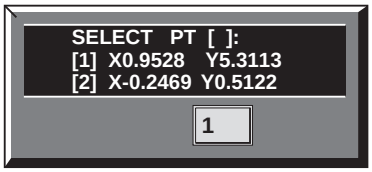


Figure D-44

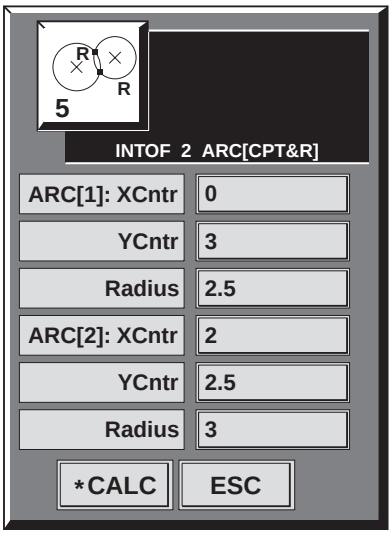


Figure D-45

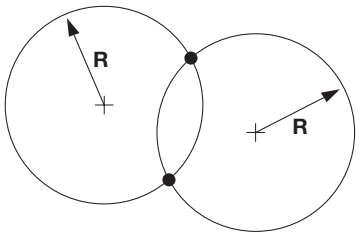


Figure D-46

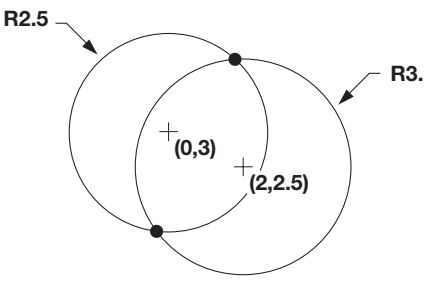


Figure D-47

5: INTOF: 2 ARCS [CNTRPT & R]

This command finds the intersection points of two arcs, each defined by its centerpoint and radius.

ARC[1]:

XCntr This is the X coordinate of the first arc's centerpoint.

YCntr This is the Y coordinate of the first arc's centerpoint.

Radius This is the radius value of the first arc.

ARC[2]:

XCntr This is the X coordinate of the second arc's center.

YCntr This is the Y coordinate of the second arc's center.

Radius This is the radius value of the second arc.

When all of the data is entered, the system calculates the two intersection points of the two arcs. The coordinates of the two points are displayed as shown in Figure D-48. One of these points can be chosen as a location in the programmed instruction. To select one of the two points, enter the number of the desired point, (either **1** or **2**) in the box shown below the points in Figure D-48. The chosen point coordinates are entered automatically into the correct data fields, when the **Enter** key is pressed. If the calculated points are incorrect, press **0** then press **Enter**.

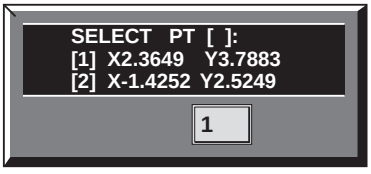


Figure D-48

4: CNTRPT: ARC [THRU 2 PTS w ANGL BETWEEN PTS]

This command calculates the centerpoint location of an arc, defined by two points on the arc, and the angle between them. Figure D-49 shows how the arc is defined.

NOTE: The angle must be less than or equal to 180°.

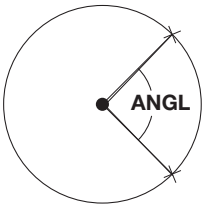


Figure D-49

NOTE: The direction of the arc must be specified as one of the parameters in this command. The direction is determined by moving around the arc from point 1 to point 2. The direction of this movement determines the direction of the arc. An example of this is shown in Figure D-50.

ARC: [PT1]:

X This is the X coordinate of the first point of the arc.

Y This is the Y coordinate of the first point of the arc.

[PT2]:

X This is the X coordinate of the second point of the arc.

Y This is the Y coordinate of the second point of the arc.

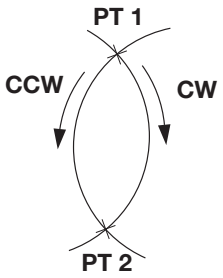


Figure D-50

Select DIR

This is the direction of the arc.

ANGLE

This is the angle between the two points.

When all of the data is entered, the system calculates the centerpoint of the arc. The coordinates of the point are automatically placed in the correct data fields of the command from which **/GEO** was selected.

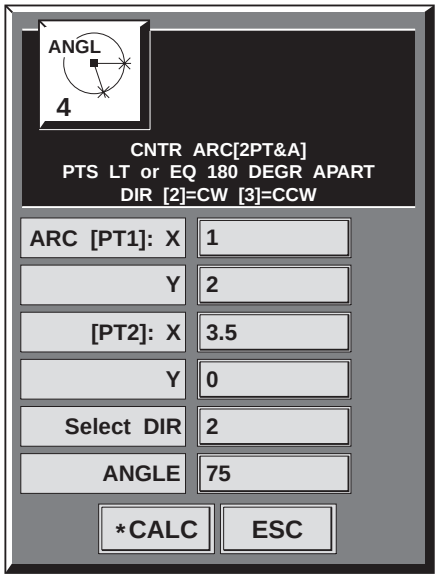


Figure D-51

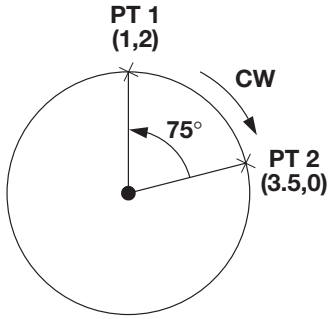


Figure D-52

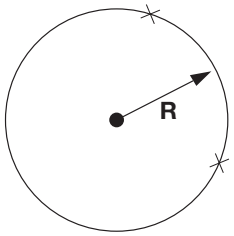


Figure D-53

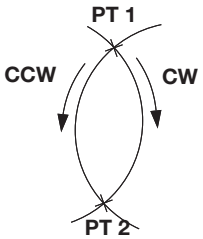


Figure D-54

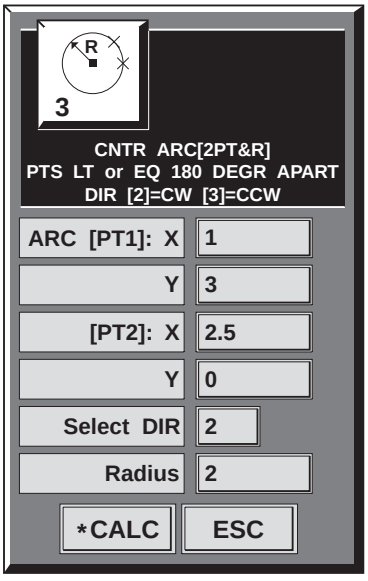


Figure D-55

3: CNTRPT: ARC [THRU 2 PTS & R]

This command finds the centerpoint location of an arc defined by two points on the arc, and the radius value of the arc. Figure D-53 shows how the arc is defined.

NOTE: The two points on the arc must be less that 180° apart.

NOTE: The direction of the arc must be specified as one of the parameters in this command. The direction is determined by moving around the arc from point 1 to point 2. The direction of this movement determines the direction of the arc. An example of this is shown in Figure D-54.

ARC: [PT1]:

X This is the X coordinate of the first point of the arc.

Y This is the Y coordinate of the first point of the arc.

[PT2]:

X This is the X coordinate of the second point of the arc.

Y This is the Y coordinate of the second point of the arc.

Select DIR This is the direction of the arc.

Radius This is the radius of the arc.

When all of the data is entered, the system calculates the centerpoint of the arc. The coordinates of the point are automatically placed in the correct data fields, when the **Enter** key is pressed.

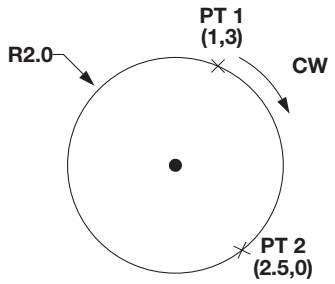


Figure D-56

2: CNTRPT: ARC [THRU 3 PTS]

This command finds the centerpoint of an arc defined with three points.

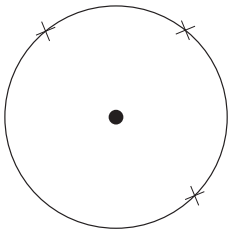


Figure D-57

ARC: [PT1]:

X This is the X coordinate of the first point of the arc.

Y This is the Y coordinate of the first point of the arc.

[PT2]:

X This is the X coordinate of the second point of the arc.

Y This is the Y coordinate of the second point of the arc.

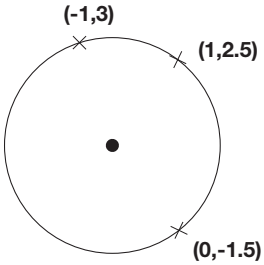


Figure D-58

[PT3]:

X This is the X coordinate of the third point of the arc.

Y This is the Y coordinate of the third point of the arc.

When all of the data is entered, the system calculates the centerpoint of the arc. The coordinates of the point are automatically placed in the correct data fields, when the **Enter** key is pressed.

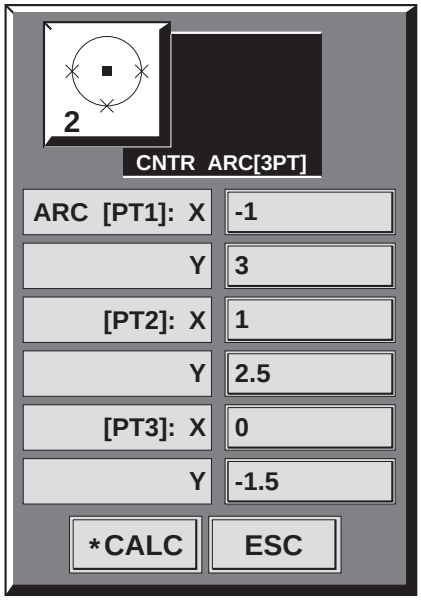


Figure D-59

1: POLAR: R A XC YC

This command calculates the X Y coordinates for a point that is specified using polar coordinates from a specified pole location. Figure D-60 shows how the polar coordinates are defined.

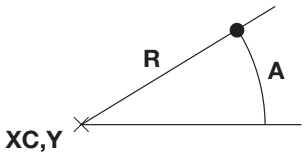


Figure D-60

NOTE: The coordinates of the pole location (XC, YC) must be given in XY coordinates.

Angle This is the angle of rotation from the X axis. This is shown in Figure E-60 as the angle marked **A**.

Radius This is the radius of the arc.

XCntr This is the X coordinate of the pole location.

YCntr This is the Y coordinate of the pole location.

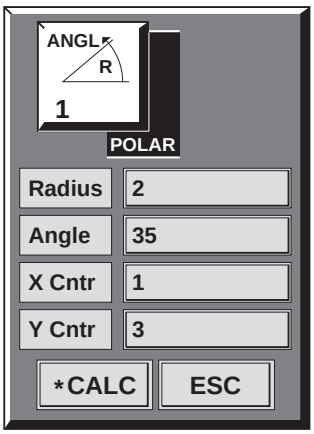


Figure D-61

When all of the data is entered, and **Enter** is pressed, the system calculates the XY coordinates of the point. The coordinates of the point are entered automatically into the correct data fields of the command.

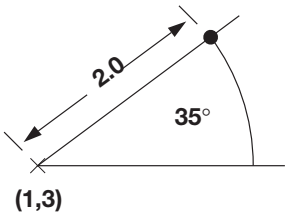


Figure D-62

0: MIDPT| LN[THRU 2PTS]

This command calculates the midpoint of a line which is given by its two endpoints. The coordinates of the two endpoints of the line must be entered, and the EZTRAK® finds the mid-point of the line. Figure D-63, below shows an example of the mid-point command.

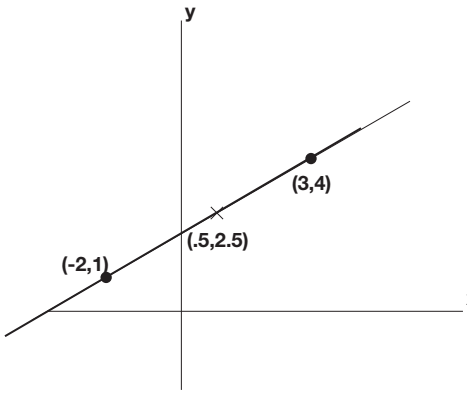


Figure D-63

ESC Pressing the **ESC** key exits the GEOMETRY HELP menu, and returns to the command in which the **/GEO** key was pressed.

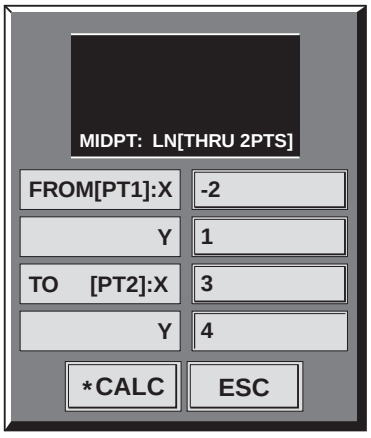


Figure D-64

• RIGHT TRIANGLE

This command computes the sides and angles of a right triangle, given a minimum amount of information. It requires at least two sides or one side and one angle to compute the remainder of the information. When sufficient data is entered, and ENTER is pressed, the missing information will appear in the empty fields. If the user simply presses ENTER again for each of the boxes, the values of the Side a and Side b are automatically entered into the X and Y endpoints of the command.

For example, if the length of **Side a** is entered as 4" and **Side b** as 3", the length of **Side c** and **Angles A and B** will be calculated automatically.

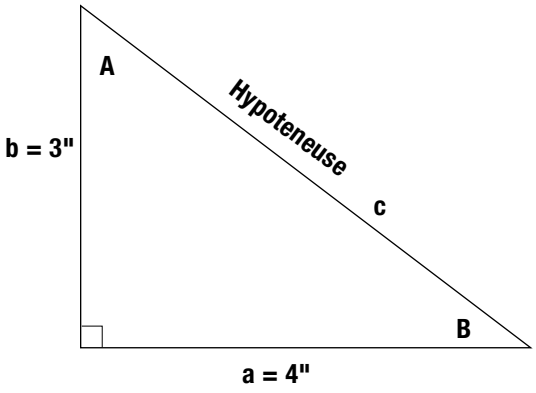


Figure D-65. When you know the lengths of two sides of a right triangle, you can use the Right Triangle function to solve for the two angles.

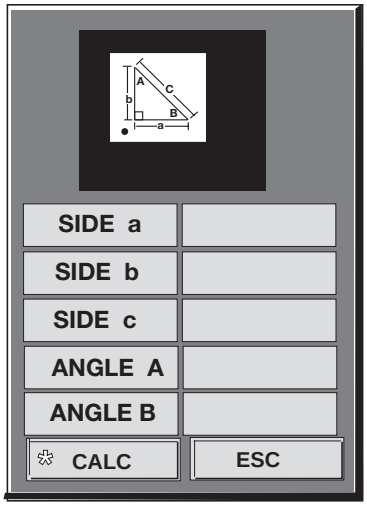


Figure D-66

Appendix E
FEED & SPEED CALCULATOR

INTRODUCTION

This section discusses the use of the **FEEDS/SPEEDS** functions that are available in many of the DO EVENT and MDI commands.

Calling up the FEEDS/SPEEDS screen

The **FEEDS / SPEEDS CALCULATOR** screen (Figure E-1) is called up when the **F6** key is pressed while entering data into a DO EVENT or MDI milling command.

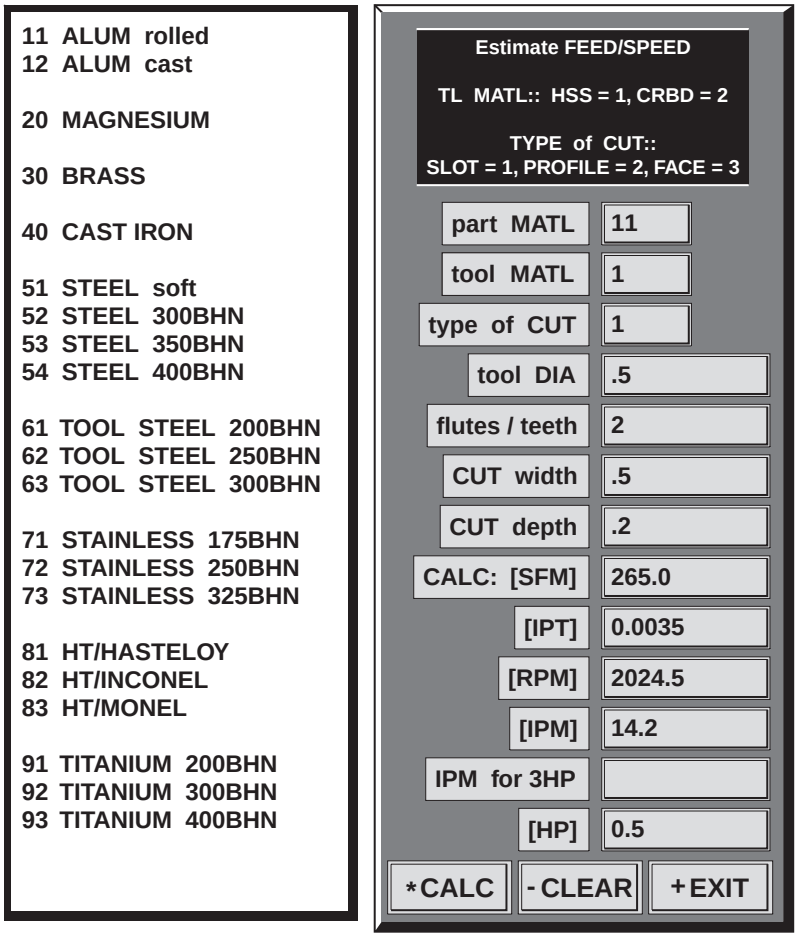


Figure E-1

Using the FEEDS/SPEEDS Calculator

The FEEDS/SPEEDS calculator is provided as a quick way to estimate the necessary spindle speed, and feedrate for any machining instruction that is programmed on the EZTRAK®.

NOTE: The values calculated by the FEEDS/SPEEDS calculator are only a starting point. Depending on the work material, the sharpness of the tool, and other variables, the calculated feedrates may not be appropriate for every application. Always double check the calculated spindle speed and feedrate before proceeding with the machining instruction.

Entering the FEEDS/SPEEDS data

The FEEDS/SPEEDS calculator requires some information before it can calculate the spindle speed and feedrate.

part MATL — This is the type of material that is being cut. Find the material type, or a material with a similar hardness value in the list at the left. Enter the number here.

tool MATL — This is the kind of tool being used. This is either **High-Speed Steel** (HSS) or **Carbide** (CRBD). Enter **1** if you are using an HSS tool, **2** if it is CRBD.

type of CUT — This is the kind of cut being made, either **SLOT**, **PROFILE**, or **FACE**. Enter **1** for a SLOT, **2** for a PROFILE, or **3** for a FACE.

tool DIA — Enter the tool diameter here.

flutes / teeth — Enter the number of flutes, or teeth on the tool here.

CUT width Enter the width of the cut here. If it is a **SLOT** cut, then enter the tool diameter here also.

CUT depth Enter the depth of the cut here.

When all of the data is entered, the calculator automatically calculates the feedrate, and spindle speed. The calculated values are shown in the lower half of the FEEDS/SPEEDS window.

CALC: **[SFM]** Surface Feet per Minute - This shows the feedrate relative to the surface of the part. (SMM - Surface Meters per Minute in metric)

[IPT] Inches Per Tooth - This shows the amount of material cut by each tooth of the tool as it rotates, and cuts the material at the given feedrate and spindle speed. (MMPT - Millimeters per Tooth in metric)

[RPM] Revolutions Per Minute - This is the calculated spindle speed for the programmed data. This is the speed which the operator is prompted to set before the instruction is executed.

[IPM] Inches Per Minute - This is the feedrate entered into the program line. (MMPM - Millimeters per minute in metric)

[IPM for 3HP] Inches Per Minute for 3 Horsepower - this is the maximum feedrate that the EZTRAK® DX can use to cut the programmed instruction given the entered values. At this feedrate, the spindle motor would be operating at its maximum horsepower rating. (MMPM for MAX - Millimeters per minute for maximum power in metric)

[HP] Horsepower - This is the approximate horsepower necessary to cut the programmed instruction at the IPM feedrate. (kW - Kilowatt necessary to cut the programmed instruction at the MMPM feedrate in metric)

In several instructions, more than one feedrate can be programmed. The pocket commands, and the COMPON move can be programmed to make a finishing pass after the initial roughing has been completed. These instructions can include a second feedrate for producing a smoother finish than the roughing operation created. The Feeds & Speeds Calculator can be used to calculate both feedrate values.

NOTE: Using the combination of a carbide tool bit, and a slot cut will cause the Feeds & Speeds calculator to return a feedrate of zero (0.0), since this is considered unacceptable machining practice.

Appendix F
ERROR MESSAGES

INTRODUCTION
Following is a list of messages that can appear on the EZTRAK screen, along with explanations of when the messages can occur and what they mean.

Messages

A:\\ SYS.BEZ not opened

The system parameter file could not be found on drive A:\. Make sure that an EZTRAK system disk is in drive A:\ whenever changing system parameters. This file contains the current software revision level, the max travel limit in X, Y, Z, the X, Y, Z user offsets, the X, Y backlash values, and the X, Y lead screw compensations.

A:\\SYS.DAT not opened

This file contains machine type (2 axis, 3 axis new style, 3 axis new style) ballscrew type (Imperial, Metric), the last part program being worked with, and the last measurement (inch or metric) being worked in. This message is typically displayed when a new EZTRAK systems disk has been sent out.

ABORT EVENT

This message is displayed when the escape key is pressed during a DO EVENT mill or drill operation.

ACTV

This word appears in the run mode message window and refers to the currently executing GCODE block. The GCODE text string for this block will appear nest to ACTV in the center run mode message window.

ALARM
HIT [ANYKEY] TO CLEAR

This message appears when the machine enters into an alarm state. A message describing the alarm condition will also appear beside this message. The operator can clear the alarm and remove the message from the display be striking any key.

ALL AXES HOMED

This message is displayed if the homing operation completed successfully.

All Parameters Set

This message informs the operator that all machine parameters in the BMDC card have been set to the values in BMDCPRMS.SYS.

Attempt to write to a write-protected disk

An attempt was made to write a part program to a write-protected floppy disk. Check that the button on the back of the disk is set to the write position.

AXES NOT HOMED--HIT [MOVE ABS]

This message is displayed if the axes have not been homed.

BLEND_CRCARC| SEQ

The blend radius on a BL ARC block is negative or zero.

|BLEND LN/ARC| SEQ

The M ARC block immediately following a BL LN block has a zero or negative radius.

|BLEND_LNLN| SEQ

The BL LN block immediate following a BL LN block cause the cutter path to re-trace the BL LN cutter path.

BLEND| SEQ

The PGM block following a BLEND LN causes the cutter path to re-trace the BL LN cutter path.

BMDC ON [V3.00]

This message displays the software revision level of the binary executive software. It appears immediately after boot-up when first entering EZTRAK or when entering EZTRAK from UTILITIES.

Cannot Open Pram File

This message is displayed just after the EZTRAK has been powered on. It informs the operator that the BMDCPRMS.SYS file does not exist on the hard drive or RAM drive.

CAN'T DO MOVE WITH POWER OFF

This message should never be displayed. The only time it is displayed is when the axis drives are down.

CAN'T SELECT INCH/METRIC LINE

This message is displayed when editing the tool table file. It appears if the operator attempts to revise the first line.

CAN'T SELECT HEADER LINE

This message is displayed when editing the tool table file. It appears if the operator attempts to revise the line containing the TNO, TLO, and DIA headings (line 2).

CAN'T SELECT NUM EQ 0

This message is displayed whenever "3 GOTO N", "6 SET N", "9 ERASE", or "5 COPY" has been selected from EDIT and the operator has selected the very first line in the program. It is only a header line that identifies the PGM and contains information that can not be executed. If it were moved, the BMDC would do a reset program every time it encountered that header line. If it were removed, the BMDC would not know the measurement mode for the program (inch or metric).

Can't SET DNC while in RUN. Hit <any KEY> to CONTINUE

This message is displayed if the operator attempts to enter DNC mode during part program execution.

CHANGE TO TOOL tool-number

This message appears inside a TEMP.TXT file whenever a tool change MCODE is encountered.

|CIRCCNTR| SEQ

This message applies to a PGM block in a part program that contains a radius value. It appears when the arc center is the same as the arc endpoint or the arc radius is zero.

||CHAMFER| SEQ

The block immediately following a chamfer block causes the spindle to re-trace the chamfer cutter path.

CLR ALL TLO/DIAS: (1 = Y/0 = N)

This message appears when "2 CLR ALL" has been selected from "* TOOLS" in "5 EDIT". It prompts the operator for confirmation before zeroing out all TLO and DIAs in the BMDC.

>> CLR SAVED PTS

This message is displayed in the TEACH mode message window in the lower left corner of the screen (just below the large coordinate window) when the operator selects "1 CLRPTS" from the TEACH button panel.

CNTR [3PTS ON ARC]
ENTER NUM of 1stPT on ARC
ENTER NUM of 2ndPT on ARC
ENTER NUM of 3rdPT on ARC

This message is displayed when "-> CRCNTR" is selected in TEACH mode. The CRCNTR function calculates the center of an arc defined by 3 points. The operator is prompted to enter the point numbers of the points in the saved points list that define the arc.

CNTR [3PTS ON ARC]: Xx-value Yy-value Rradius-value
ENTER NUM TO SAVEPT or [.] =MOVETO PT

These messages are displayed in TEACH mode when "-> CRCNTR" is selected from the save points button panel and after supplying the point numbers of the points that define the arc. The X and Y coordinates of the calculated arc center and radius are displayed and the operator is prompted to either enter the point number in the save points list to save it, or enter " . " to move the spindle to the arc center point coordinates.

COM ERROR: COMM PORT 1 NOT SET

This message is displayed when there is a loose cable connection to port 1 or when the connection has not been established.

>>COMM PORT1 9600,8,NO,1 || OPEN

This message appears when a successful link has been established. MODEMs with baud rates up to 9600 bits per second can be used.

Communication Error
Remote Cancel

This message is displayed when the communications link between EZTRAK and a remote terminal was cancelled by the remote terminal.

COPY disk in FLOPPY DRIVE

This message appears when the operator elects to do a complete disk copy. This message lets him know that a complete disk copy is in progress.

Could not locate sample MW font

EZTRAK uses special fonts to draw the graphics characters such as up and down arrows. These graphics characters are taken from font files located on the EZTRAK system disk. This message is displayed if any of the following font files are not present on the system diskette:

HLV050B.FN_
HLV025B.FN_
BLD019M.FN_
BLD019B.FN_
BLD013.FN_
BLD015.FN_

CYCLE DEFINED INCORRECTLY

This message is displayed when an edit field has been left blank. Make sure that all data entry fields have numbers in them.

DISPLAY LIST not opened

The DISPLAY.LST file could not be opened. This temporary file contains the point to point locations used by "* VIEW PART" function to draw the part. Make sure there is memory available on the C:\ drive.

DIVISION BY ZERO ERROR

This message appears in "* CALC" when an expression in the denominator of a division operation evaluates to zero.

DNC internal error -- ABORTED
Hit any key to continue

This message is displayed while running under DNC mode. DNC mode is used when the size of the part program to run is larger than 256Kbytes. The operator receives this message because there was an error detected while writing a portion of the DNC file to the BMDC. Check the BMDC board.

DNC file error - ABORTED
The format of the DNC file is not compatible with the IBM PC file format.

DNC OFF? <Y>
or <ESC>=CANCEL
This message prompts the operator for confirmation before turning DNC off.

DO EVENT
This message appears in the large mode window located in the upper right corner of the display when DO EVENT mode is active.

>>DO MDI EVENT
This message applies to DO EVENT mode. It appears just before a DO EVENT move is to be executed.

>>DO MOVE
This message is displayed in the upper right corner of the "3 MOV XYZ" display. It lets the operator know that he is in MOVE XYZ mode.

DONE RECEIVING FILE PGM-file-name
Notifies the operator the receive operation has completed successfully.

DONE SENDING FILE file-name
This message notifies the operator that the send operation has completed successfully.

Drive not ready
A floppy disk could not be found in drive A:\. Check that there is a floppy disk in the disk drive, that the disk is pressed in, and that the drive door is closed all the way.

DRIVES NOT ON, FAILED TO GO HOME!
This message is displayed while attempting to home the axes with power off.

END OF PROGRAM
This message is displayed while previewing a part program when the end of the program has been reached. It informs the operator that preview has completed.

—— END OF PROGRAM || Hit any key to EXIT
This message applies when "–" is selected to view TEMP.TXT in RUN mode. It appears when the end of TEMP.TXT has been reached.

[ENTER]FUNC:
This message is displayed whenever a trig function is selected from "* CALC". It prompts the operator to enter an expression for the trig function. Note that after the expression is typed in, the operator need not press the " > " key for the closing parenthesis. Pressing the enter key automatically adds the closing parenthesis and enters the result into the desired data entry field.

ENTER [0] TO CLR PTS
ENTER [ ] to CANCEL

These messages are displayed when the operator selects "1CLRPTS" from the TEACH mode button panel. The operator is prompted to confirm or cancel the operation. A clear points operation can be cancelled by leaving the data entry area blank or confirmed by entering in a zero (0).

ENTER TOOLNO
"0" = SET GAUGE HEIGHT
"ESC" = CANCEL
CURRENT GAUGE HEIGHT = current-gauge-height
ENTER GAUGE HEIGHT

This message is displayed whenever "2 TLO = Z" is selected from the JOG menu. It prompts the operator to enter the tool number of the tool whose TLO will be set equal to the Z coordinate. If a "0" is entered, the operator is allowed to change the gauge height. The current gauge height setting in inches (or mm) is displayed following the "GAUGE HEIGHT = " message. Initially, it will be set to zero. The gauge height added to the TLO will become the new Z coordinate.

( ) ERR

This message is displayed in "* CALC" when the number of parentheses on the left of an arithmetic expression is different from the number of parentheses on the right of the expression.

ERROR: BREAK TRAP NOT SET

This message is displayed when " < 7 > SEND or RECEIVE files" has been selected, followed by " < 3 > ASCII" or " < 2 > YMODEM" and a link could not be established between EZTRAK and a remote terminal. Specifically interrupt signal handling (ISH) could not be set. ISH allows a send or receive operation to be aborted by hitting the escape key.

ERROR: Cannot Save Received File

The file that was received from a remote machine could not be saved on drive C:\. Make sure that drive C:\ can be written to.

ERROR: DESIGNATED FILE [PGM file-name] NOT FOUND
ERROR: DESIGNATED FILE [TXT file-name] NOT FOUND

The file to be sent could not be found on EZTRAK.

ERROR error-number LOADING file-name

The PGM file could not be loaded because one of the following errors (listed by error number) has occurred:

1. File is too big.
2. BMDC error.
3. BMDC write error.

ERROR — EXPRESSION MUST BE POSITIVE

This message appears in "* CALC" when an expression inside a square root function evaluates to a negative number.

ERROR in Reading DISK!! Hit <any KEY>

This message is displayed when:

1. TEMP.TXT GCODE file could not be created from the DNC PGM file for some reason. Check available hard disk space.
2. The TXT file to DNC would not be found.

ERROR IN SENDING FILE

An error was encountered while attempting to send a file to a remote device.

ERROR — INPUT FIELD EMPTY: CHECK DATA

This message appears whenever a data entry edit field was left blank.

Error loading sample MW font
Error loading sample MW font 1
Error loading sample MW font B
Error loading sample MW font X
Error loading sample MW font 0

EZTRAK uses special fonts to draw the graphics characters such as up and down arrows. These graphics characters are taken from font files located on the EZTRAK system disk. These messages are displayed if:

1) There is not enough program memory to store the font file contents.
2) One or more of the sectors on drive C:\ used to store the information in the font file are bad.

ERROR – NO POINT AT point-number

This message applies when ".MOVPTS" has been selected from the TEACH button panel and the number of the point to move to has just been entered. It appears when there is no points list entry at the specified point number.

ERROR opening COMP.TMP

The temporary file COMP.TMP is a storage area that will contain the cutter compensation GCODE for the second pass of a COMP|RPT block. Check that the sectors on the C:\ drive that contain COMP.TMP can be written to.

ERROR opening PSURF.LST

This file contains the point locations used by preview to draw the part. Make sure there is available program memory and available memory on the C:\ drive.

ERROR opening PGM-file-name

The PGM file to edit could not be opened. Check available memory on the C:\ drive, check that the C:\ drive can be written to, and check that sectors on the C:\ drive containing the PGM file can be written to.

ERROR opening PGM-file-name

This message is displayed when the PGM file to be loaded could not be opened. Check available memory on the C drive.

ERROR opening PRGM PGM-name

The selected PGM program could not be found.

ERROR opening PRGM TEMP.TXT

No GCODE file could be found for the PGM program.

ERROR opening TEMP.TXT

This message applies when "–" is selected to view the TEMP.TXT file in RUN mode. It appears when the TEMP.TXT file could not be found.

ERROR opening TXT FILE TXT-file-name

This message is displayed when the selected TXT file could not be found.

ERROR — PGM LOAD FAILED LOADING file-name

The PGM file could not be loaded because of a BMDC error.

Error Reading Param File

This message is displayed if the machine parameter file (BMDCPRMS.SYS) has been corrupted for some reason (bad sector). Run the update program. If the problem still occurs:

1. Hook up a full keyboard
2. Type: ren bmdcprms.sys *.bad
3. Insert your backup disk in drive A:\
4. Type: copy A:\BMDCPRMS.SYS C:\
5. Remove full keyboard and re-attach the EZTRAK keyboard
6. Type: 1 and press ENTER to return to EZTRAK

ERROR RECEIVING FILE PGM-file-name

An error was encountered while attempting to receive a file from a remote device.

ERROR — |UNEXPECTED EOF| SEQ

This message is displayed if a blend line or blend arc is the last PGM block in the program.

expr value ERR

This message appears in "* CALC" when ENTER was pressed without entering a value.

EXIT EZTRAK

This message is displayed whenever the operator exits EZTRAK into Utilities.

EXIT EZTRAK UTILITIES

This message appears when the operator escapes back into EZTRAK from UTILITIES.

EXIT ON MOVE DONE

This message is displayed in BLOCK run mode when the operator selects "0 EXIT" while the axes are still in motion.

EXIT TO FILES UTILITIES

This message appears when 9 UTILS is selected from the main button panel. It informs the operator that EZTRAK is being exited and UTILITIES is being entered.

FAULT: EZ-LINK communications aborted

This message is displayed when the operator selects "9 UTILS" from EZTRAK and then the "4 COMM" key from Utilities. A link could not be established between EZTRAK and a remote terminal.

File Write Error

The operator receives this message during a send or receive operation in "9 UTILS". EZTRAK could not write the data for the file being sent to the destination machine or could not write the data for the file being received to the receiving machine.

File Read Error

This message appears in "9 UTILS" if an I/O error occurs while sending or receiving a file. Check the disk sectors being used for the file being sent or received on the sending machine if you are sending a file and on the remote machine if you are receiving a file.

FIND SEQNO sequence-number

This message appears when the operator selects "3 FND SEQ" from RUN mode showing the sequence number to search for in the part program.

FIND TOOLNO tool-number

This message appears when the operator selects "3 FND SEQ" from RUN mode showing the tool number to search for in the part program.

||Hit [1] to CONTINUE or [0] to EXIT

This message is displayed when "–" is selected to view the TEMP.TXT file in RUN mode and the TEMP.TXT file is too large to fit on one page. The maximum number of GCODE lines that will fit on a page is displayed and the operator is given the option to review the remainder of the file or exit.

HIT HOLD BEFORE EXIT

This message is displayed in AUTO run mode when the operator selects "0 EXIT" while the axes are still in motion.

HIT HOLD BEFORE RESET

This message appears during RUN mode when the operator attempts to reset the program while the axes are still in motion.

HOLD feedrate-override-percentage

This message is displayed when the machine is in a HOLD state. It appears in the PGM status window on the right side of the display next to the feedrate override percentage.

>>HOME AXES

This prompt appears in the upper right hand corner of the display while the axes are being homed.

HOMING | TIMED-OUT

This message is displayed if it took longer than 60 seconds to home all of the axes.

HOMING | FAILED: HIT KEY TO CONTINUE

This message is displayed if a BMDC failure was detected while attempting to home the axes.

HP INCORRECTLY CALCULATED

This message applies to a feeds and speeds calculation. The value entered for the part material or the tool material is unknown.

|ILLEGAL LAST MOVE IN COMP| SEQ
sequence-number [HIT KEY to CONTINUE]

This message is displayed if COMP ON, BL LIN, or BL ARC is the last PGM block in the program.

>>INPUT DATA

This prompt appears in the prompt window in the upper right corner of the display during any data entry operation. It informs the operator that he is in data entry mode.

INPUT LESS THAN 2 HOLES

This message appears when "7 DR ROW", "8 DR BOX" or "9 DR BC" has been selected and the number of holes to drill is less than 2. The minimum number of holes to drill for any of these selections is always 2.

INTOF 2 LNS
ENTER NUM of 1stPT on 1stLN
ENTER NUM of 2ndPT on 1stLN
ENTER NUM of 1stPT on 2ndLN
ENTER NUM of 2ndPT on 2ndLN

These messages are displayed when " < - INTOF" is selected (computes the intersection point of 2 lines) in TEACH mode. The operator is prompted to enter the point numbers of the 2 points that define the first line and the point numbers of the 2 points that define the second line.

INSERT OLD SYSTEM DISK
TO TRANSFER PARAMETERS
HIT ANY KEY TO CONTINUE

This message is displayed when the machine parameter disk file could not be found on the hard drive. It informs the operator to insert the backup disk in drive A:\ to transfer the machine parameter file from the backup disk to the hard drive or RAM.

INSERT NEW SYSTEM DISK
TO COPY SYS PARAMETERS
HIT ANY KEY TO CONTINUE

This message only applies to 2-axis machines with RAM drives. It informs the operator to insert the system disk in drive A:\ to copy the machine parameters from the RAM drive to the system disk. It will use these parameter values whenever the machine is powered up.

INTOF 2LNS: Xx-value Yy-value
ENTER NUM TO SAVEPT or [.] =MOVETO PT

These messages apply during TEACH mode when the operator selects " < - INTOF". They appear after the operator enters in the point numbers of the 2 intersecting lines. The X and Y coordinates of the calculated intersection point is displayed and the operator is prompted to enter the point number in the save points list to save it, or enter " . " to move the spindle to the intersection point coordinates.

>>JOG X
[+][–] = FEED OVR

This message informs the operator that the axes are set to jog in the X direction and that pressing the " + " and "–" keys increases or decreases the feed override percentage, respectively.

>>JOG Y
[+][–] = FEED OVR

This message informs the operator that the axes are set to jog in the Y direction and that pressing the " + " and "–" keys increases or decreases the feed override percentage, respectively.

>>JOG Z
[+][–] = FEED OVR

This message informs the operator that the axes are set to jog in the Z direction and that pressing the " + " and "–" keys increases or decreases the feed override percentage, respectively.

LAST SAVED PT point-number

This message is displayed in the mode message window (in the lower left corner of the display) whenever a point is entered into the points list window. It lets the operator know the point number of the last saved point. Up to 100 points can be saved in the points list.

LINES ARE PARALLEL

This message applies to " < - INTOF" in TEACH mode. The lines defined by the selected saved point numbers never cross.

MOVE ABS Xx-coordinate Yy-coordinate Zz-coordinate
MOVE INC Xx-coordinate Yy-coordinate Zz-coordinate

This message appears in the center of the "3 MOV XYZ" display. It displays the X, Y and Z coordinates to move to, the move mode (ABS or INC), and a " + MOVE" and an ESC button. The operator presses " + " to execute the move or escape to abort the move.

MOVE DONE

This message is displayed whenever a DO EVENT move is completed.

MOVE TO CLEARANCE POINT

This message is displayed when **F1 MOVE TO CLR PT** has been selected from **3 MOV XYZ**. It informs the operator that the EZTRAK® is about to move to a pre-set clearance point. To set a clearance point, select **SET XYZ** from the main or **JOG** menu and then select F1.**NEXT**
This word appears in the run mode message window and refers to the very next GCODE block to be executed. The GCODE test string for this block will appear next to NEXT in the bottom run mode message window.

MOVETO PT
ENTER PT [ ]

This message is displayed in TEACH mode when ".MOVPTS" is selected from the save points mode button panel. The operator is prompted to enter the point number of the point to move the spindle to.

MOVETO X=x-coordinate Y=y-coordinate
[.] = MOVETO PT

This message is displayed when ".MOVPTS" is selected from the TEACH button panel and the number of the point to move has just been entered. The X and Y coordinates of the point to move to is displayed and the operator is prompted to confirm the move. The move can be aborted by leaving the edit field blank.

NEW PTS FILE

This message is displayed in TEACH mode whenever the points list is empty. The points list can be empty when the operator first enters TEACH mode or immediately following the execution of a clear points command.

NO FILES FOUND *.PGM
NO FILES FOUND *.TXT

This message appears if, after the root directory on the A:\ or C:\ drive has been searched for PGM or TXT files, no files were found having the "PGM" or "TXT" extensions.

NO MEMORY AVAILABLE

The machine ran out of program memory while attempting to display the file selection list for EDIT or LOAD.

NOTE: COPY will involve swapping the original disk and the blank disk 3 times.

This message informs the operator of what the disk copy operation involves.

NOTE – KEY INVALID WITH COMP ON

This message applies when a PGM program is edited or created and the line being edited or created is between COMP ON and COMP OFF. Only M LINE, M ARC, BL LIN, and BL ARC moves are allowed. This message will be displayed if any other key is pressed.

Nsequence-number ;SUBPRGM subprogram-name

This message notifies the operator that the selected sub-program is being loaded into memory.

NUMBER OF HOLES MUST BE A WHOLE NUMBER

This message is displayed while attempting to execute a DRILL ROW, DRILL RECT, or DRILL BC operation in DO EVENT or MDI. It appears when the number of holes entered is not a whole number.

OLD SYS PARAMETERS NOT FOUND
DEFAULT VALUES WILL BE USED
HIT ANY KEY TO CONTINUE

This message is displayed when the machine parameter file could not be found on the backup disk. The BMDC will be loaded with default values.

OPERATION ABORTED

This message is displayed while in PREVIEW mode. It appears if a key is pressed during a drawing operation. The drawing operation is aborted and the function associated with the key pressed is executed.

OPERATOR STOP

The operator receives this message when an optional stop (||STOP) has been encountered while previewing a part program and the operator selects "0" to STOP preview (instead of " + " to continue previewing).

OUT OF MEMORY

This message is displayed when the machine runs out of program memory while attempting to insert a PGM line into the file being edited.

PLEASE ENTER 1 FOR MILL AND 0 FOR DRILL

This message is displayed in TEACH mode immediately before the calculated intersection or circle center point is to be saved in the points list. The operator is prompted to enter the point type (mill or drill) to save it as.

PLEASE ENTER 2 FOR CW OR 3 FOR CCW

A value other than 2 for CW or 3 for CCW was entered for a M ARC or circle mill operation.

Please move Z axis below travel limit before entering 3-axis mode

This message is displayed while attemptiing to enter 3-axis mode from 2-axis mode. It appears when the Z axis is on or above the home switch. To satisfy this message, move the Z axis down until you hear a click.

POWER/OFF MANUAL

This message appears in the large mode window located in the upper right corner of the display. It is displayed whenever the axis drives are off.

POWER/ON

This message appears in the large mode window located in the upper right corner of the screen. It is displayed whenever the axis drives are on.

POWER/ON AUTO RUN

This message is displayed when the machine is in AUTO mode and the axes are in motion. In AUTO mode, the machine executes the part program non-stop from start to finish.

POWER/ON AUTO RDY

This message is displayed when the machine is in AUTO mode and the axes are not in motion due to an auxiliary function. In AUTO mode, the machine executes the part program non-stop from start to finish.

POWER/ON BLOCK

This message is displayed in the large mode window in the upper right corner during RUN, when the machine is running in BLOCK mode. In BLOCK mode, the machine pauses after executing each PGM block.

[POWER ON] FAILED

The operator receives this message when EZTRAK could not turn power on.

POWER/ON SET RUN

This message appears in the upper right corner of the display when the operator first enter RUN mode. It prompts the operator to select the mode to run in (AUTO BLOCK).

PREV

This word appears in the run mode message window and refers to the previously executed GCODE block. The GCODE test string for this block will appear next to **PREV** in the top run mode message window.

PRGM pgm-name LOADED

This message states that the specified part program has been successfully loaded into the BMDC card to be executed.

PRGM LOAD FAILED

The operator receives this message when one of the following three errors occurred:

1. TEMP.TXT file could not be opened. Make sure there is available memory on drive C:\.
2. One of the following errors was encountered while attempting to write TEMP.TXT to the BMDC card:
    a) File is too big.
    b) BMDC error
    c) BMDC write error
3. An error was found while attempting to generate GCODE for the PGM file. Check the PGM file for a programming error.

PROGRAM STOP

This message is displayed when an optional stop (||STOP) has been encountered in a part program while it is being previewed. The operator is then given the option of continuing the part program or making a function key selection.

PROGRAM ERR

This message is displayed when an alarm occurred while previewing a part program. A typical alarm situation would be moving outside the maximum travel limits, for example.

RADIUS MUST BE POSITIVE

A negative radius value was entered for a DO EVENT or MDI move.

RAPID

This message appears when the axes are in motion during a rapid move.

rdParamFile(parameter-number) Failed
Error Setting Parameter parameter-number

This message is displayed if the front end software failed to set a machine parameter in the BMDC to a specific value or the BMDC got hung up. Check the BMDC board.

Read fault

An error was detected while reading from a floppy or hard drive. Run a diagnostic test on the floppy or hard drive.

Reading Param File...

This message warns the operator not to attempt an operation (such as manual axis movement) until all machine parameters have been set.

READING PGM-file-name

Notifies the operator that the selected PGM is being loaded into memory.

Reading Tool Table File...

This message is displayed whenever "*TOOLS" is selected from "5 EDIT". It informs the operator that the stored TLO and DIA values are being loaded from hard disk to the BMDC card.

Receive file terminated

This message notifies the operator that we are exiting the receive operation.

>>SAVE PTS [JOG X]

This message is displayed when TEACH mode is entered by first selecting "1 JOG X" from the main button panel and then selecting ".SAVPTS".

>>SAVE PTS [JOG X]

This message is displayed in the mode message window (just under the large coordinate window) when ". SAVPTS" is selected from the JOG button panel.

>> SAVED PT point-number

This message appears during TEACH mode immediately after a mill or drill point is entered into the points list. It displays the point number in the points list that the mill or drill point was saved under.

STEP = .0005

This message is displayed in JOG mode and shows the step increment to jog in inches or millimeters. The axes are moved by the step increment each time the "- > " or " < -" keys are pressed. Pressing "- > " will step .0005 inches in the positive X or Y direction (depending on whether JOG X or JOG Y was selected) and " < -" will step .0005 in the negative X or Y direction.

SCANNING PATH... *.PGM
SCANNING PATH... *.TXT

This message appears immediately after the operator selects "5 EDIT" from the main button panel or " < - LOAD" from RUN mode. The root directory on the C:\ or A:\ drive (depending on whether "F5 change to drive A:" has been selected) is searched for all file names having the "PGM" or "TXT" extensions (depending on whether "F4 MORE FILES" has been selected). After the file search completes, the operator is presented with a list of files to choose from. This message informs the operator to wait until the file search on the A:\ or C:\ drive has completed.

SET AXES | FAILED

This message is displayed if the BMDC failed to SET X, Y, Z.

SET AXES | TIMED-OUT

This message is displayed when a SET X, Y, Z command was submitted and the BMDC timed out. This could occur immediately after homing the machine or during SET X, Y, Z.

>>SELECT EVENT

This message appears when the machine is in DO EVENT mode. It prompts the operator to select a DO EVENT function.

SELECT DNC MODE:
1 = RUN FROM DISK
2 = LOAD DNC MACROS

This prompt is displayed whenever the operator enters DNC mode. If RUN FROM DISK is selected, the operator will be presented with a load menu to select the file to DNC. After the selection has been made, the first 8 K bytes of the selected file will be loaded into the DNC area of BMDC memory. If LOAD DNC MACROS is selected, the operator will be presented with a load menu to select the file containing the macros the DNC program will use. After the selection has been made, the DNC macros file will be loaded into the BMDC memory.

Select DNC FILENAME

This message tells the operator that a file will now be selected to DNC

>>SELECT FUNCTION

This message prompts the operator to select a function. It appears in the prompt window in the upper right corner of the display and is present whenever the main button panel is displayed.

| SEQNUM NOT FOUND — END OF FILE |

This message appears when GOTO N has been selected and the requested sequence number has not been found. Note that GOTO N only searches from the current cursor position to the end of the file. If you wish to find a sequence number before the cursor, move the cursor to the top of the file before selecting GOTO N.

SET OFFSETS

This message is displayed immediately after the X, Y, Z coordinates are entered for "7 SET X, Y, Z". It lets the operator know that the coordinates for the current spindle position are being changed.

SET X = 0
[PART PROGRAM COORDINATE SYSTEM]

This message is displayed when SET X = 0 is selected from the jog button panel. This function allows the operator to make the X user coordinate value for the current spindle position zero.

SET Y = 0
[PART PROGRAM COORDINATE SYSTEM]

This message is displayed when SET Y = 0 is selected from the jog button panel. This function allows the operator to make the Y user coordinate value for the current spindle position zero.

SET Z = 0
[PART PROGRAM COORDINATE SYSTEM]

This message is displayed when SET Z = 0 is selected from the jog button panel. This function allows the operator to make the Z user coordinate value for the current spindle position zero.

syntax ERR

This message is displayed when "* CALC" is selected from a data entry box in DO EVENT or MDI. It appears when an invalid calculation was typed in. An example of this would be typing in "7*" (without the number to the right of the multiplier operator) as a calculation.

START| PGM-name

This comment appears inside a TEMP.TXT file. It is always the first line in the file and identifies the PGM by it's number.

STEP OVER LT .005

This message is displayed when:

1. The pocket step over for milling a rectangular pocket is less than .005 inches.
2. The Y-step over for a face milling operation is less than .005 inches.
3. The pocket step over for milling a circular pocket is less than .005 inches.

TEMP.TXT not opened

This message appears after selecting " + DO PTS" from ".SAVPTS". The TEMP.TXT file (contains the GCODE for the TEACH.PGM) could not be opened. Make sure there is available memory on drive C:\.

TEACH.PGM not opened

The TEACH program that will contain the PGM for the saved points could not be opened. Make sure there is available memory on the C:\ drive.

3 PTS ARE CO-LINEAR

This message applies to "- > CRCNTR" in TEACH mode. The selected saved points defining the arc form a straight line.

to copy all EZTRAK files
from floppy disk to EZTRAK, Hit <+>:
from EZTRAK to floppy disk, Hit <+>:

These messages appear in 9 UTILS upon selecting " < 2 > COPY files from EZTRAK to floppy disk" or " < 3 > COPY files from floppy disk to EZTRAK" and then selecting " < 5 > COPY all EZTRAK files from EZTRAK to FLOPPY disk" or < 6 > COPY all EZTRAK files from FLOPPY disk to EZTRAK". It prompts the operator for confirmation. The operation can be aborted by hitting a key other than " + ".

to COPY ALL OTHER files
from EZTRAK to floppy disk, Hit <+>:
from floppy disk to EZTRAK, Hit <+>:

These messages appear in 9 UTILS upon selecting " < 2 > COPY files from EZTRAK to floppy disk" or " < 3 > COPY files from floppy disk to EZTRAK" and then selecting " < 5 > COPY other files from EZTRAK to FLOPPY disk" or " < 6 > COPY other files from FLOPPY disk to EZTRAK". It prompts the operator for confirmation. The operation can be aborted by hitting a key other than " + ".

TO DISPLAY PGM ON FLOPPY DISK
Type PGM NAME and press ENTER
[NO NAME to EXIT]

This message is displayed when 9 UTILS is selected followed by " < 5 > VIEW contents of file on FLOPPY DISK" and then " < 2 > View a EZTRAK file on FLOPPY DISK". The operator is prompted to enter the name of a PGM file on the A:\ drive to view.

TO DISPLAY TXT ON FLOPPY DISK
Type TXT NAME and press ENTER
[NO NAME to EXIT]:

This message is displayed when 9 UTILS is selected followed by " < 5 > VIEW contents of file on FLOPPY DISK" and then " < 4 > View OTHER file on FLOPPY DISK". The operator is prompted to enter the name of a TXT file on the A:\ drive to view.

TO DISPLAY TXT on EZTRAK
Type TXT NAME and press ENTER
[NO NAME to EXIT]

This message is displayed when 9 UTILS is selected followed by " < 6 > VIEW contents of file on EZTRAK" and then " < 2 > View OTHER file on EZTRAK". The operator is prompted to enter the name of a TXT file on the C:\ drive to view.

TOOL CHANGE

This message appears at the bottom of the preview window whenever a tool change PGM statement has been encountered. Preview pauses to allow the operator to change the tool diameter, re-size the preview window, etc.

[TURN ON SPINDLE TO ENABLE CYCLE START]

The operator will receive this message if he tries to perform a cutting operation with the spindle off.

TURN ON SPINDLE TO ENABLE PROGRAM START

The operator will receive this message if he tries to execute a program with the spindle off.

User Cancel

The operator receives this message after "9 UTILS" has been selected and a communications link has been established. It is displayed when the operator hits escape during a send or receive operation.

>>USE JOG KNOB = .1/REV OR SEL SW

This message is displayed in jog mode. It informs the operator to turn the +/- Jog Select switch on the front panel to the direction to jog then push the +/- Jog Select switch to jog the selected axis and to use the rotary switch (job knob) to move in finer increments (.1 inches per revolution).

VIEW PART
[0]=EXIT [1]=REVIEW

This message appears whenever part geometry is being displayed. Select "0" to remove the view geometry display from the screen. Select "1" to re-display the part program geometry or to view a different section of the part program.

>> WAIT [CYCLE]

This message appears when the axes are in motion or during a Z positioning move. It informs the operator to wait until the cycle completes before selecting another function.

WAIT...
DELETE: Type PGM NAME and press ENTER
[No Name to EXIT]

This message is displayed when 9 UTILS is selected followed by "<4> DELETE FILES" and then "<7> DELETE A EZTRAK FILE from EZTRAK and FLOPPY DISK". The operator is prompted to enter the name of a PGM file to delete.

WAIT...
TO DISPLAY PGM on EZTRAK
Type PGM NAME and press ENTER
[NO NAME to EXIT]:

This message is displayed when 9 UTILS is selected followed by "<6> VIEW contents of file on EZTRAK" and then "<2> View and EZTRAK file on EZTRAK". The operator is prompted to enter the name of a PGM file on the C:\ drive to view.

WAIT [MOVE]

This message is displayed while the spindle is moving to a saved point in TEACH mode after selecting ".MOVPTS". It informs the operator to wait until the move completes before making another key selection.

WAIT [MOVE]

This message appears in the top right corner of the "3 MOV XYZ" display. It tells the operator to wait until the move finishes executing before selecting another function.

WAIT — SENDING protocol-type FILE [file-name] — HIT [ESC] TO ABORT

Where protocol type can be:

1. EZ-LINK – the communication format used by Bridgeport equipment.
2. YMODEM – a type of communication protocol.
3. ASCII – sends an ASCII file straight through without any modifications.

A send operation is in progress. To abort the operation, hit the escape key.

WAIT -- SAVING CLEARANCE POINT

This message is displayed after the clearance values have been entered after selecting **F1 SET CLR PT** from **SET XYZ**. It informs the operator to perform no operation until the clearance point values have been saved to disk.

Wait -- QUILL UP

This message can appear when "4 QU UP" has been selected from the Main menu or when "/ QU UP" is selected from JOG or DO EVENT. It warns the operator not to attempt any other operation until the Z axis is in the Quill Up (home) position.

Waiting for BMDC -- countdown

This message is displayed if the binary executive has not finished loading. If countdown reaches zero, the binary executive did not load properly. Make sure that the EZLOAD.EXE and BMDC.BIN files exist on the hard drive or system disk (two axis machines without hard drives only).

WARNING||
WRITE PROTECT the original disk before copying.

This message appears as a safety precaution to protect the original disk before the copy operation takes place.

WARNING — GEOM.LST NOT CREATED

GEOM.LST is a file containing point locations used by "VIEW PART" to draw the part. Make sure that there is available memory on the C:\ drive.

WARNING — PGM-file-name NOT SAVED

A PGM file could not be saved to the C:\ drive.

WARNING — PGM-subprogram-file-name NOT SAVED

The subprogram called by the part program does not exist.

WARNING — PGM EXISTS . . [0]=OVERWRITE

A copy of the PGM file to be saved already exists on the drive where the file is being saved.

WARNING — TEXT NOT COPIED

The temporary file COPY.TMP used to hold the PGM lines to be copied could not be opened. Check for available memory on the C:\ drive and check that the sectors on the C:\ drive containing the COPY.TMP file can be written to.

WARNING — TEXT NOT PASTED

The temporary file COPY.TMP used to hold the PGM lines to be pasted could not be opened. Check for available memory on the C:\ drive and check that the sectors on the C:\ drive containing the COPY.TMP file can be written to.

Write fault

An error was detected while writing to a floppy or hard drive. Run a diagnostic test on the floppy or hard drive.

Wrong Param File Version Rev revision-level required.

This message is displayed when the he machine parameter file (BMDCPRMS.SYS) is not compatible with the current front end software. Run the update program to make the parameter file current.

Z DEPTH LT .0001

Certain canned cycles require a Z depth. They are:

1. Mill Rectangle
2. Mill Rectangular Pocket
3. Face Mill
4. Slot Mill
5. Circle Mill

Appendix G

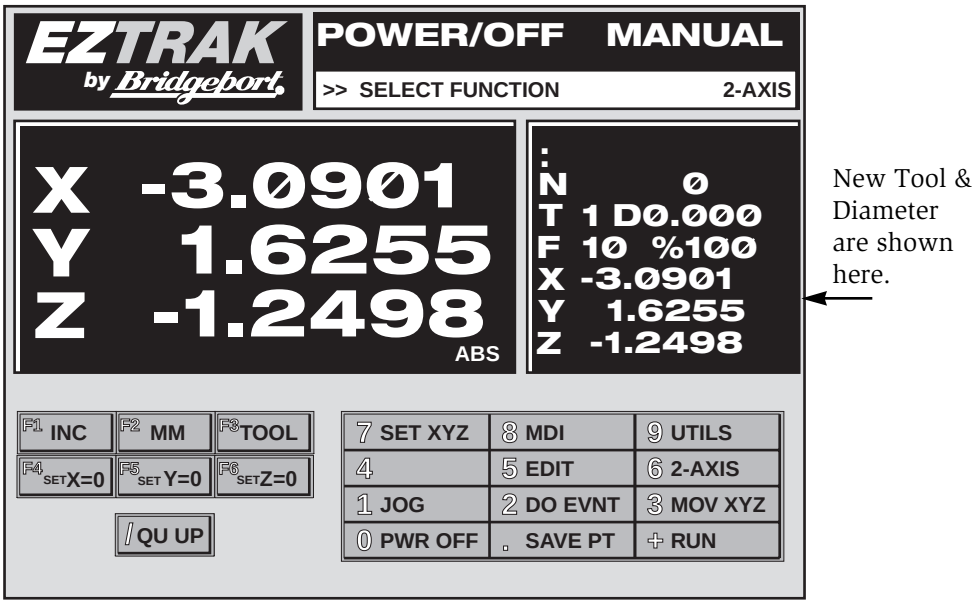
CREATING A TOOL & SETTING A CLEARANCE POINT

G.1 INTRODUCTION

This section presents procedures which can be used to manage Tools (i.e., create, delete, change) and set clearance points on the EZTRAK 3-Axis milling machine.

Creating a Tool:

To create new Tools on Bridgeport Machines' EZTRAK milling machine, begin at the Main Screen as shown in Figure G-1.



Fligure G-1: Main Operating Screen.

1. Press the **F3** key (**F3 TOOL**) which opens the dialog box shown in Figure G-2. Type a "0" to create a new Tool, then press **ENTER**.

2. Figure G-3 illustrates the next dialog box that comes up which prompts you to input a number for the new Tool, then press **ENTER**.

3. You will now see a new dialog box (Figure G-4) that contains your new Tool number. Next you should type in the diameter of the new Tool and press **ENTER**.

4. The Main Operating Screen will now be displayed, and will reflect the new Tool number which you just created.

The EZTRAK is capable of creating and storing up to 24 Tools in the TLO Library.

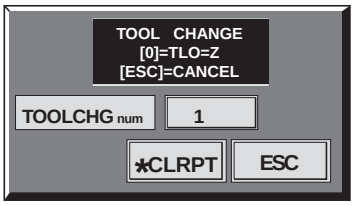


Figure G-2: Enter a Tool Number (1-24) to change Tools or 0 to create a new Tool. Escape to cancel the operation.

Figure G-3

Figure G-4: TNO should equal your new Tool number. This box prompts you for the diameter of the Tool.

G.2 TOOL LIBRARY

Tool attributes such as the Tool Number (TNO), Tool Length Offset (TLO) and Diameter (DIA) are stored in the Tool Library. Workshifts are also listed in the Tool Library and may be accessed for editing or modification.

To access the Tool Library from the Main Operating Screen, press the **5** key (**5 EDIT**). Figure G-5 will be displayed. By selecting the * key (* **TOOLS**), you can access the tool file in the list box shown as Figure G-6. You can then access the tool and workshift lists in Figure G-7.

At this screen, you can do the following:

- Use the arrow keys to move the cursor within the list in order to select the desired tool or workshift.

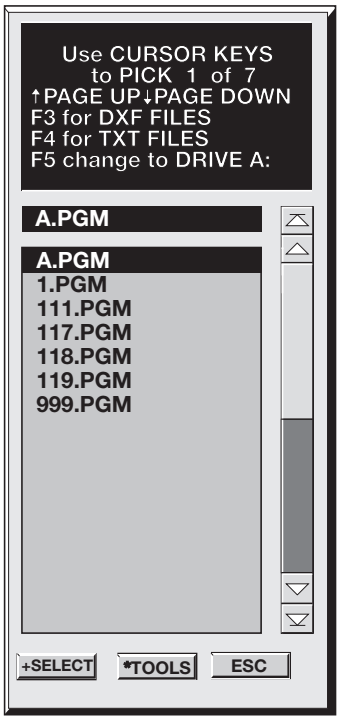


Figure G-5: Press * to access the Tool Library.

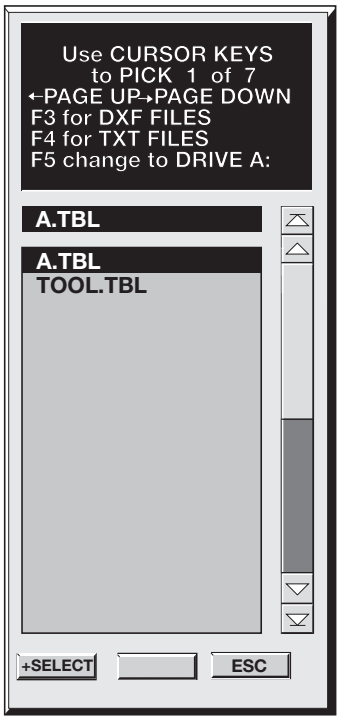


Figure G-6

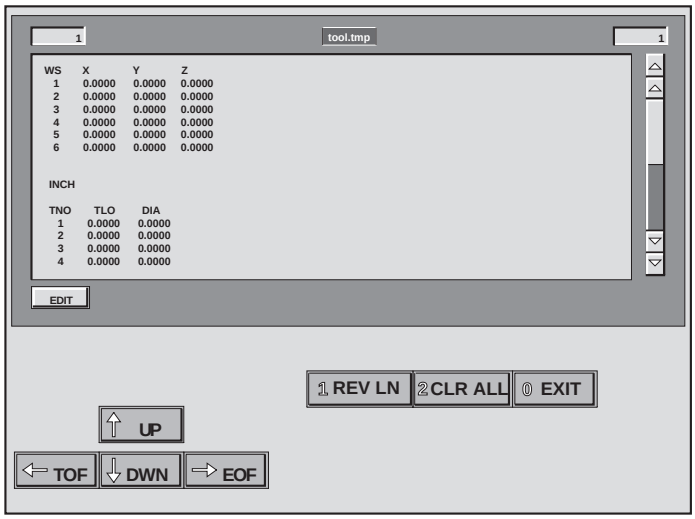


Figure G-7: Tool Library

- Use the **1** key to revise values for tools or workshifts.

- Use the **2** key to clear all the Tools from the library (see Figure G-10); and

- Use the **0** key to exit the Tool Library.

A box will come up (see Figure G-10) allowing you to save changes that have been made. Press the + key (**+ Save**) to save to disk, or the * key (*SAVE/LOAD) to save and load to memory.

For further information on the Tool Library, see the* **Tools** section in Chapter 8 of this manual.

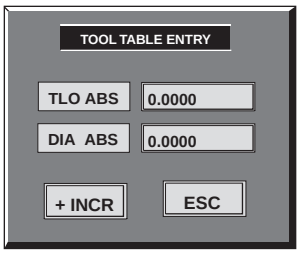


Figure G-8

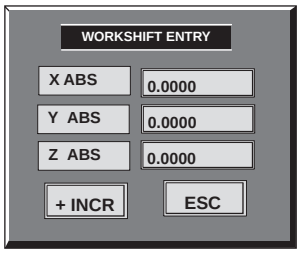


Figure G-9: Edit workshift coordinates

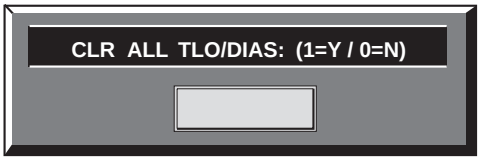


Figure G -10: Enter Y To clear all Tools from the Library.

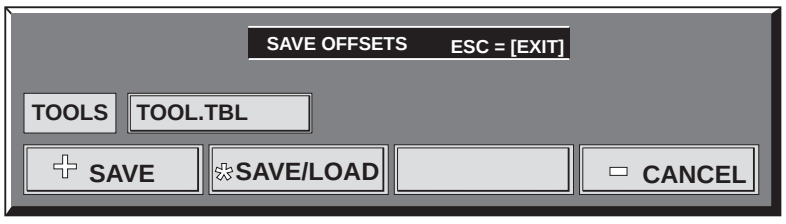


Figure G-11

G.3 CHANGING TOOLS

To make a Tool change, press the **F3** key (**F3 TOOLS**) from the Main Screen. You will see the same screen used when creating a Tool shown in Figure G-2. Type in the Tool number you wish to make active, and press the **ENTER** key.

The Tool you just created should now be shown on the Main Screen (Figure G-1).

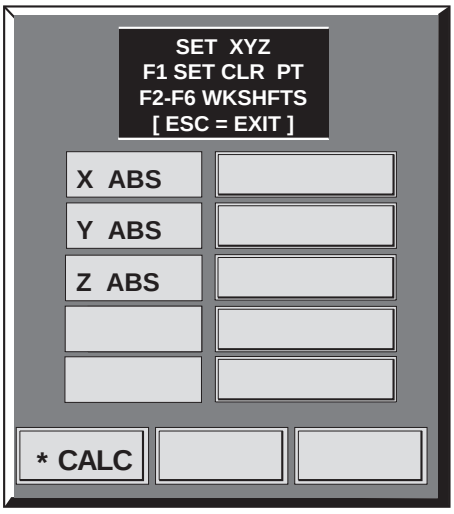


Figure G-12

G.4 SETTING A CLEARANCE POINT

The following procedure describes how to set a Clearance Point. A Clearance Point is a location for the EZTRAK table to be moved to where there will be enough space for the operator to "safely" conduct a Tool change, OR to change a part after being milled. First, physically move the Tool to a location away from the part to be milled by moving the EZTRAK table using the Jog keys or the **3** key (**3 MOVE ABS**). Next, from the Main Screen, press the **7** key (**7 SET XYZ**). Figure G-12 will be displayed. Press the **F1** key to set the Clearance Point. A message will flash "WAIT...SAVING CLEARANCE POINT".

Once the Clearance Point is set, you can move to that Clearance Point for any Tool from either the Main Screen or the Jog Screen.

Figure G-13

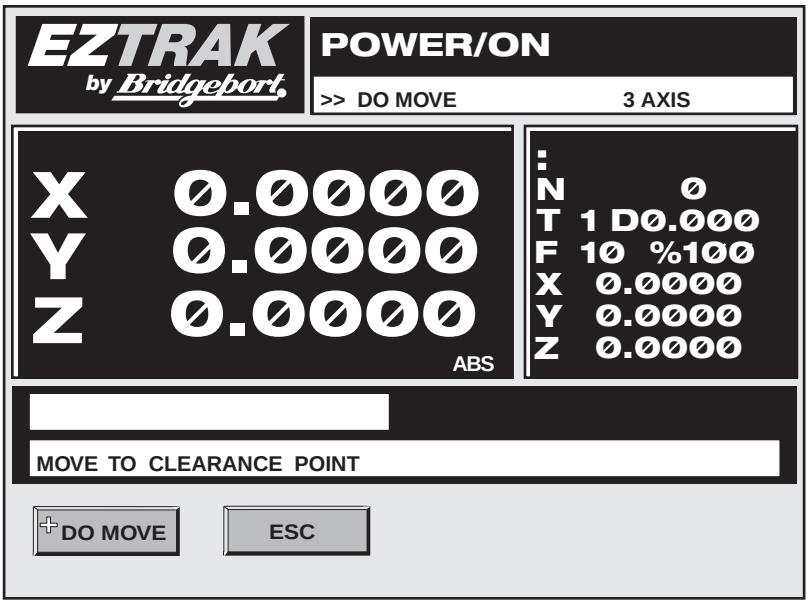


Figure G-14

- **from the Main Screen:**

  Press the **3** key (**3 MOVE XYZ**) to get to the Figure G-13 dialog box; OR

- **From the Jog Screen:**

  Press the **3** key (**3 MOVE XYZ**) to get Figure G-13.

By pressing the **F1** key (**MOVE TO CLR PT**), the Figure G-14 dialog box will appear prompting you to press the + key (**+ DO MOVE**) which will move the Tool to the Clearance Point (i.e., table will move so that the Tool will be positioned at the Clearance Point).

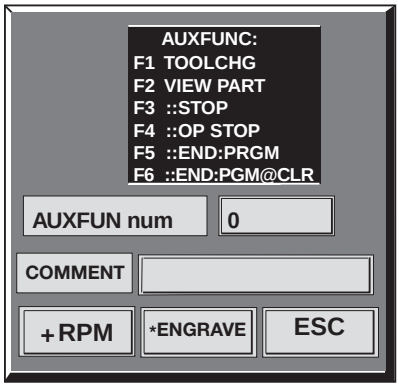


Figure G-15: To access this dialog box, press the + key (+ PGMSTOP) from the MDI Screen.

It is also possible to program a Tool change line at a Clearance Point in the MDI Mode.

The following procedure outlines how to create a Tool change line.

1. In MDI, press the + key (**+ PGM STOP**) to access the dialog box shown in Figure G-15.

2. Press the **F1** key for the Tool change, then Figure G-16 will be displayed.

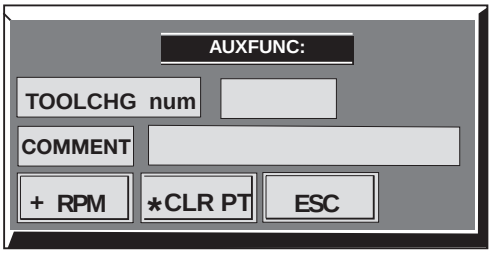


Figure G-16

3. Press the * key (*** CLR PT**) to access the dialog box shown in Figure G-17. Type the number of the Tool which will be changed at the Clearance Point then press **Enter**.

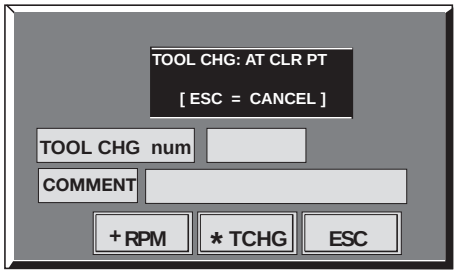


Figure G-17

This will create a new instruction line in the program which will look like this:

```
0010¦¦  TOOLCHG ¦CLRPT  T3
```

In the above example, the EZTRAK table will move to the Clearance Point so the operator can change to Tool #3.

To end a program at a Clearance Point, from the MDI Screen, press the + key (**+ PGM STOP**), then the **F6** key. The program line will look like this.

```
0030¦¦END ¦ PRGM@CLR
```

G.5 SAVING TOOLS

To enter tools into the Tool Library, the following procedure is necessary:

1) In the **BASIC OPERATIONS** screen press the **1 JOG** button.

2) In the **JOG** screen press the **2 TLO = Z** button.

3) A dialogue box will appear as in Figure G-18 below.

Figure G-18

Enter a Tool Number (1- 24) that corresponds to the new tool being input. If a Tool Number is not entered, the operator has the choice of pressing ESC to CANCEL the operation.

4) If a Tool Number is entered, the following dialogue box will appear.

Figure G-19

Enter the Tool Diameter, then press ENTER. The machine automatically calculates the TLO for the tool based on how far the tool tip is from zero at the maximum Quill Up (top dead center) position.

INDEX

A

B

C

D

E-F

G-H

I-K

***

***

L-M

***

N-P

***

Q-R

S-T

U-W

X-Z