

TRAK[®] DPMV5, DPMV7 ProtoTRAK[®] VM CNC

Safety, Programming, Operating & Care Manual

Document: P/N 24491 Version: 031705
--



Southwestern Industries, Inc.

2615 Homestead Place

Rancho Dominguez, CA 90224-5610 USA

T | 310.608.4422 | F | 310.764.2668

Service Department: 800.367.3165

e-mail: sales@southwesternindustries.com | service@southwesternindustries.com | [web: southwesternindustries.com](http://web:southwesternindustries.com)

Copyright 2005, Southwestern Industries, Inc. All rights are reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, mechanical, photocopying, recording or otherwise, without the prior written permission of Southwestern Industries, Inc.

While every effort has been made to include all the information required for the purposes of this guide, Southwestern Industries, Inc. assumes no responsibility for inaccuracies or omission and accepts no liability for damages resulting from the use of the information contained in this guide.

All brand names and products are trademarks or registered trademarks of their respective holders.

Southwestern Industries, Inc.
2615 Homestead Place
Rancho Dominguez, CA 90220
Phn 310/608-4422 ♦ Fax 310/764-2668
Service Department
Phn 800/367-3165 ♦ Fax 310/886-8029

Table of Contents

1.0	Introduction		5.0	Definitions, Terms & Concepts	
1.1	Manual Organization	1	5.1	ProtoTRAK VM Axis Conventions	25
2.0	Safety		5.2	Part Geometry & Tool Path Programming	25
2.1	Safety Publications	3	5.3	Planes and Vertical Planes	26
2.2	Danger, Warning, Caution and Note Labels and Notices Used in this Manual	3	5.4	Absolute & Incremental Reference	26
2.3	Safety Precautions	6	5.5	Referenced & Non-Referenced Data	27
3.0	Description		5.6	Incremental Reference Position in Programming	27
3.1	Display Pendant Front	9	5.7	Tool Diameter Compensation	27
3.1.1	Keyboard Hard Keys	9	5.8	Tool Diameter Compensation When Contouring in Z with Part Geometry	29
3.1.2	Soft Keys	10	5.9	Connective Events	30
3.1.3	Emergency Stop Switch	11	5.10	Conrad	30
3.1.4	The Liquid Crystal Display	11	5.11	Memory & Storage	31
3.2	Pendant Left Side	11	6.0	DRO Mode	
3.3	Pendant Right Side	12	6.1	Enter DRO Mode	33
3.4	Machine Specifications	13	6.2	DRO Functions	33
3.5	Spindle Motor	15	6.3	Jog	34
3.6	Electronic Handwheels	15	6.4	Power Feed	34
3.7	Auxiliary Functions	15	6.5	Do One	35
3.8	Limit Switches	15	6.6	Go To	35
3.9	Lubrication System	16	6.7	Teach	35
3.9.1	Factory Default Values	16	6.7.1	Entering Teach Data	35
3.10	Electrical Cabinet	16	6.8	Return to Absolute Zero	36
3.11	Optional Equipment	16	6.9	Spindle	36
3.11.1	Position Encoders	16	6.9.1	Spindle Speed	36
3.11.2	Power Draw Bar	16	6.9.2	Spindle Hard Keys	36
3.11.3	Remote Stop Go Switch	16	6.10	Tool #	37
3.11.4	Work Light	17	7.0	Program Mode	
3.11.5	Coolant Pump	17	Part 1: Getting Started & Some General Info		
3.11.6	Spray Coolant	17	7.1	Programming Overview	39
3.11.7	Chip Pan/Splash Shield	17	7.2	Enter Program Mode	39
3.11.8	Table Guard	17	7.3	Program Header Screen	40
3.11.9	DXF File Converter	17	7.3.1	Program Name	40
4.0	Basic Operation		7.3.2	General Program Options	41
4.1	Switching the TRAK DPM V3/V5 On/Off	19	7.3.3	Program Header Softkeys	42
4.2	Switching the ProtoTRAK VM On	19	7.4	Auxiliary (AUX) Functions	42
4.3	Shutting down the ProtoTRAK VM	20	7.5	Multiple Fixtures	43
4.4	Spindle Forward/Off/Reverse	21	7.5.1	The Default Fixture	44
4.5	Manual Operation of Ram, Table, Saddle	21	7.5.2	Fixtures & Running the Program	44
4.6	Emergency Stop	21	7.5.2	Editing Fixtures	44
4.7	Switching Between Two and Three-Axis Operation	21	7.6	Assumed Inputs	44
4.8	Spray Coolant	22	7.7	Z Rapid Positioning	45
4.9	Windows Desktop	22	7.8	Softkeys within Events	45
4.10	Help Functions	22	7.9	Programming Events	46
4.10.1	Math Helps	22	7.10	Editing Data While Programming	47
4.11	Windows Up or Down	24	7.11	LOOK	47
4.12	Using a Mouse & Keyboard	24	7.12	Finish Cuts	48
			7.13	Two Versus Three-Axis Positioning	49

8.0	Program Mode			
	Part 2: Program Events			
8.1	Posn Position Events	51		
8.2	Drill Events	51		
8.3	Bolt Hole Events	52		
8.4	Mill Events	52		
8.5	Arc Events	53		
8.6	Pocket Event	54		
	8.6.1 Circular Pocket	54		
	8.6.2 Rectangular Pocket	55		
	8.6.3 Irregular Pocket	55		
	8.6.4 Tool Path in Pocket Events	56		
	8.6.5 Zigzag Z Depth Cuts	56		
	8.6.6 Conrad in Pocket Events	57		
	8.6.7 Bottom Finish Cut	57		
8.7	Islands	57		
	8.7.1 Circular Island	57		
	8.7.2 Rectangular Island	58		
	8.7.3 Irregular Island	59		
8.8	Profile Events	60		
	8.8.1 Circle Profile	60		
	8.8.2 Rectangular Profile	61		
	8.8.3 Irregular Profile	62		
8.9	Helix Events	62		
8.10	Subroutine Events	63		
	8.10.1 Repeat	64		
	8.10.2 Mirror	65		
	8.10.3 Rotate	65		
8.11	Copy Events	65		
8.12	Thread Mill Event	66		
8.13	Pause Events	67		
8.14	Tap Events	67		
	8.14.1 DPMV Tapping Speed			
	Recommendations	67		
	8.14.2 Tapping Notes and			
	Recommendations	68		
8.15	Engrave Event	68		
	8.15.1 Prompts for the Engrave Event	69		
8.16	Finishing Teach Events	70		
9.0	Program Mode			
	Part 3: The Auto Geometry Engine			
	(A.G.E) Programming			
9.1	Starting the A.G.E.	71		
9.2	A.G.E. Mill Prompts	72		
9.3	A.G.E. Arc Prompts	73		
9.4	Skipping Over Prompts	73		
9.5	The OK/NOT OK Flag	73		
9.6	Ending A.G.E.	74		
9.7	Guessing Data	74		
9.8	Look & Guess	75		
9.9	Calculated Data	76		
9.10	Arcs and Conrads	76		
9.11	Tangency	76		
10.0	Edit Mode			
10.1	Delete Events	79		
10.2	Spreadsheet Editing™	79		
			10.2.1	Selecting Data to be Displayed on the Search Edit Table 80
			10.2.2	Sorting Data 81
			10.2.3	Making Global Changes to Data 81
		10.3		Erase Program 83
		10.4		Clipboard 83
		10.5		G-Code Editor 84
11.0	Set Up Mode			
11.1	The Tool Table	87		
	11.1.1 The Tool Table Screen	88		
	11.1.2 The Logic of the Tool Table	88		
	11.1.3 Initial Tool Set-Up	89		
	11.1.4 Starting Over: Erasing Tool Info	90		
	11.1.5 Adding a Tool	90		
	11.1.6 Replacing a Tool	90		
	11.1.7 Z Modifiers	91		
	11.1.8 Resetting the Reference Point	91		
	11.1.9 Saving Tool Information	91		
	11.1.10 Opening a Program	91		
	11.1.11 Making Tool Set-Ups Easy	91		
	11.1.12 Tool Table & 2-Axis CNC Operation	92		
11.2	Tool Path	92		
11.3	Reference Positions (REF POSN)	93		
	11.3.1 Z Retract	94		
	11.3.2 Home Positions	94		
	11.3.3 Limit Positions	94		
11.4	Fixture Offsets	94		
11.5	Service Codes	95		
12.0	Run Mode			
12.1	Run Mode Screen	97		
12.2	Two Versus Three-Axis Running	97		
12.3	Starting to Run	98		
12.4	Program Run	98		
12.5	TRAKing	99		
	12.5.1 TRAKing in Two-Axis CNC	99		
12.6	Program Run Messages	100		
12.7	Stop	100		
12.8	Feedrate Override	100		
12.9	Trial Run	100		
12.10	Data Errors	100		
12.11	Fault Messages	101		
12.12	Run Sequence	101		
13.0	Program In/Out Mode			
13.1	Softkey Selections in the Program In/Out Mode	103		
13.2	Basic Navigation of Program In/Out Mode Screens	104		
	13.2.1 Basic Parts of the Program In/Out Mode Screens	104		
	13.2.2 Softkeys in the Program In/Out Mode Screens	105		
13.3	Opening a File	105		
13.4	Saving Programs	106		
13.5	Copying Programs	107		
13.6	Deleting Programs	107		

13.7	Renaming	108
13.8	Backing Up	109
13.9	Converters™	110
13.9.1	Activating Converters	110
13.9.2	Converting from a Different Format Into a ProtoTRAK VM	112
13.9.3	Opening G-code Files	112
13.9.4	Converting from the ProtoTRAK VM to a Different Format	113
13.10	ProtoTRAK & TRAK CNC Compatibility	113
13.10.1	File Formats	114
13.10.2	Opening .MX2 & .MX3 Files on a ProtoTRAK VM CNC	114
13.10.3	Running ProtoTRAK VM Files on ProtoTRAK & TRAK CNC Controls	115
13.11	Running G Code Files	116
13.11.1	G Codes Recognized by the ProtoTRAK VM	116
13.11.2	M Codes Supported by the ProtoTRAK VM	117
13.11.3	Valid Characters for Word/ Address Sequences	117
13.12	Networking	117
13.12.1	Network Security	118
13.12.2	Peer-to-Peer Networking	118
13.12.3	Basic Network Set-Up	119
13.12.4	Server and Client Networking	124
13.12.5	Network Description of the ProtoTRAK	126
13.13	RS232 Interface	126
13.13.1	Connections	126
13.13.2	Receiving a File	127
13.13.3	Sending a File	128
13.14	CAD/CAM and Post Processors	128
13.14.1	Writing a Post Processor	129
13.14.2	Convertible G-Codes	130
13.14.3	Supported Addresses	131
13.14.4	Format Terms & Definitions	131
13.14.5	G Codes that Generate Errors	132
13.14.6	Accepted M Codes	133

Addendum 1

Sample Program No. 1	
Basic 2 ½-Axis Program	A1-i
Sample Program No. 2	
2-Axis Profile with Limited Print Data	A1-ii

ProtoTRAK VM Training Checklist

1.0 Introduction

Congratulations! Your TRAK DPM V milling machine with the ProtoTRAK VM CNC is the ultimate toolroom machine. It features an easy-to-use interface, a Windows operating system and dozens of features that maximize machinist's productivity for any kind of toolroom job.

Manual machining is always available and made easier with features like electronic handwheels, 250 inch per minute rapids, tool offsets and all the best features of sophisticated DRO's.

Two-axis machining is available at the touch of a button for prototyping and moderately complex, low volume work.

Three-axis machining is programmed and run with unprecedented flexibility. Programs may be entered at the control or imported from CAD/CAM files. Advanced color graphics show program features. TRAKing allows manual control of the programmed motion.

The operation of the ProtoTRAK VM has been painstakingly refined to bring you the best in technology while retaining the ease of use that has made ProtoTRAK the top brand in controls for low volume production.

1.1 Manual Organization

Section 2 of this manual provides important safety information. It is highly recommended that all operators of this product review this safety information.

Section 3 provides a description of the TRAK DPM V5, V7 and the PROTOTRAK VM CNC.

Section 4 describes the operation of the milling machine and some basic operations of the ProtoTRAK VM CNC.

Section 5 defines some terms and concepts useful in learning to program and operate the PROTOTRAK VM.

The PROTOTRAK VM is organized into six Modes of operation that are described in the following sections.

Section 6 DRO: Digital Readout, jog, and powerfeed operations.

Section 7 Programming, Part 1: covers some general programming information and instructions on starting new programs.

Section 8 Programming, Part 2: Program Events - instructions for the canned cycles, or events used to program the ProtoTRAK VM.

Section 9 Programming, Part 3: the A.G.E., or Auto Geometry Engine, so powerful it gets its own section.

Section 10 Edit: for routines to make large-scale changes to programs in current memory, including the powerful Spreadsheet Editing®

Section 11 Set-Up: Tool information, part graphics and special codes.

Section 12 Run: Instructions on running a program to machine your part.

Section 13 Program In/Out: Storing and managing your programs.

2.0 Safety

The safe operation of the TRAK DPM V5 or V7 depends on its proper use and the precautions taken by each operator.

- Read and study this manual. Be certain every operator understands the operation and safety requirements of this machine *before* its use.
- Always wear safety glasses and safety shoes.
- Always stop the spindle and check to ensure the CNC control is in the stop mode before changing or adjusting the tool or workpiece.
- Never wear gloves, rings, watches, long sleeves, neckties, jewelry, or other loose items when operating or around the machine.
- Use adequate point of operation safeguarding. It is the responsibility of the employer to provide and ensure point of operation safeguarding per OSHA 1910.212 - Milling Machine.

2.1 Safety Publications

Refer to and study the following publications for assistance in enhancing the safe use of this machine.

Safety Requirements For The Construction, Care And Use of Drilling, Milling, and Boring Machines (ANSI B11.8-2001). Available from The American National Standards Institute, 1430 Broadway, New York, New York 10018.

Concepts And Techniques Of Machine Safeguarding (OSHA Publication Number 3067). Available from The Publication Office - O.S.H.A., U.S. Department of Labor, 200 Constitution Avenue, NW, Washington, DC 20210.

2.2 Danger, Warning, Caution, and Note Labels and Notices As Used In This Manual


DANGER - Immediate hazards that **will** result in severe personal injury or death. Danger labels on the machine are red in color.

WARNING - Hazards or unsafe practices that *could* result in severe personal injury and/or damage to the equipment. Warning labels on the machine are orange in color.

CAUTION - Hazards or unsafe practices that *could* result in minor personal injury or equipment/product damage. Caution labels on the machine are yellow in color.

NOTE - Call attention to specific issues requiring special attention or understanding.



SOUTH WESTERN INDUSTRIES 2615 HOMESTEAD PLACE, RANCHO DOMINGUEZ, CA 90220			
MODEL	TRAK DPM V		
S/N			
ELECTRICAL RATINGS:			
220	VOLTS	30	AMPS
60	Hz	3	PHASE
FLA OF LARGEST MOTOR		13.5	AMPS
SHORT CIRCUIT INTERRUPT		200K	AMPS
ELECTRICAL DRAWING #:		21840-1	
CNC CONTROL ELECTRICAL RATINGS: IF APPLICABLE			
115	VOLTS	8	AMPS
60	Hz	1	PHASE
MACHINE (ONLY) MADE IN "XXXXXX"			

i00774

Safety & Information Labels Used On The TRAK DPM V5 or V7

It is forbidden by OSHA regulations and by law to deface, destroy or remove any of these labels



**HIGH
VOLTAGE**

230 VOLTS

ELECTRICAL HAZARD

Potential lethal voltages exist in this drive and remain at a dangerous level for several minutes after power is removed. Before attempting to service this controller, measure the DC bus voltage to ensure that it is zero.

WARNING

IMPROPER INSTALLATION OR OPERATION OF THIS MOTOR MAY CAUSE INJURY TO PERSONNEL OR MOTOR FAILURE. READ OPERATING INSTRUCTIONS.

MOTOR MUST BE INSTALLED AND GROUNDED PER LOCAL AND NATIONAL ELECTRICAL CODES.

TO REDUCE POTENTIAL OF ELECTRICAL SHOCK DISCONNECT ALL POWER SOURCES BEFORE INITIATING ANY MAINTENANCE OR REPAIRS.

KEEP FINGERS AND FOREIGN OBJECTS AWAY FROM VENTILATION AND OTHER OPENINGS. KEEP AIR PASSAGES CLEAR.

EYEBOLTS OR LIFTING HOOKS, WHEN SUPPLIED, ARE INTENDED FOR LIFTING THE PRODUCT ONLY AND MUST NOT BE USED TO LIFT ADDITIONAL WEIGHT.

USE ONLY GRADE 5 SHOULDER EYEBOLTS WITH MINIMUM THREAD LENGTH OF 1 1/2 TIMES DIAMETER.

Safety & Information Labels Used On The TRAK DPM V5 or V7

*It is forbidden by OSHA regulations and by law to deface, destroy or remove any
of these labels*

2.3 Safety Precautions

1. Do not operate this machine before the **TRAK DPM V5 or V7 Safety, Installation, Maintenance, Service and Parts List Manual**, and the **TRAK DPM V5 or V7 Safety, Programming, Operating & Care Manual** have been studied and understood.
2. Do not run this machine without knowing the function of every control key, button, knob, or handle. Ask your supervisor or a qualified instructor for help when needed.
3. Protect your eyes. Wear approved safety glasses (with side shields) at all times.
4. Don't get caught in moving parts. Before operating this machine remove all jewelry including watches and rings, neckties, and any loose-fitting clothing.
5. Keep your hair away from moving parts. Wear adequate safety headgear.
6. Protect your feet. Wear safety shoes with oil-resistant, anti-skid soles, and steel toes.
7. Take off gloves before you start the machine. Gloves are easily caught in moving parts.
8. Remove all tools (wrenches, check keys, etc.) from the machine before you start. Loose items can become dangerous flying projectiles.
9. Never operate a milling machine after consuming alcoholic beverages, or taking strong medication, or while using non-prescription drugs.
10. Protect your hands. Stop the machine spindle and ensure that the CNC control is in the stop mode:
 - Before changing tools
 - Before changing parts
 - Before you clear away the chips, oil or coolant. Always use a chip scraper or brush
 - Before you make an adjustment to the part, fixture, coolant nozzle or take measurements
 - Before you open safeguards (protective shields, etc.). Never reach for the part, tool, or fixture around a safeguard.
11. Protect your eyes and the machine as well. Don't use a compressed air hose to remove the chips or clean the machine (oil, coolant, etc.).
12. Stop and disconnect the machine before you change belts, pulley, gears.
13. Keep work area well lighted. Ask for additional light if needed.

14. Do not lean on the machine while it is running.
15. Prevent slippage. Keep the work area dry and clean. Remove the chips, oil, coolant and obstacles of any kind around the machine.
16. Avoid getting pinched in places where the table, saddle or spindle head create "pinch points" while in motion.
17. Securely clamp and properly locate the workpiece in the vise, on the table, or in the fixture. Use stop blocks to prevent objects from flying loose. Use proper holding clamping attachments and position them clear of the tool path.
18. Use correct cutting parameters (speed, feed, depth, and width of cut) in order to prevent tool breakage.
19. Use proper cutting tools for the job. Pay attention to the rotation of the spindle: Left hand tool for counterclockwise rotation of spindle, and right hand tool for clockwise rotation of spindle.
20. Prevent damage to the workpiece or the cutting tool. Never start the machine (including the rotation of the spindle) if the tool is in contact with the part.
21. Check the direction (+ or -) of movement of the table when using the jog or power feed.
22. Don't use dull or damaged cutting tools. They break easily and become airborne. Inspect the sharpness of the edges, and the integrity of cutting tools and their holders. Use proper length for the tool.
23. Large overhang on cutting tools when not required result in accidents and damaged parts.
24. Prevent fires. When machining certain materials (magnesium, etc.) the chips and dust are highly flammable. Obtain special instruction from your supervisor before machining these materials.
25. Prevent fires. Keep flammable materials and fluids away from the machine and hot, flying chips.
26. When working in manual mode (not CNC) make sure the computer control is switched to DRO or OFF.

3.0 Description

3.1 Display Pendant Front (see Figure 3.1)

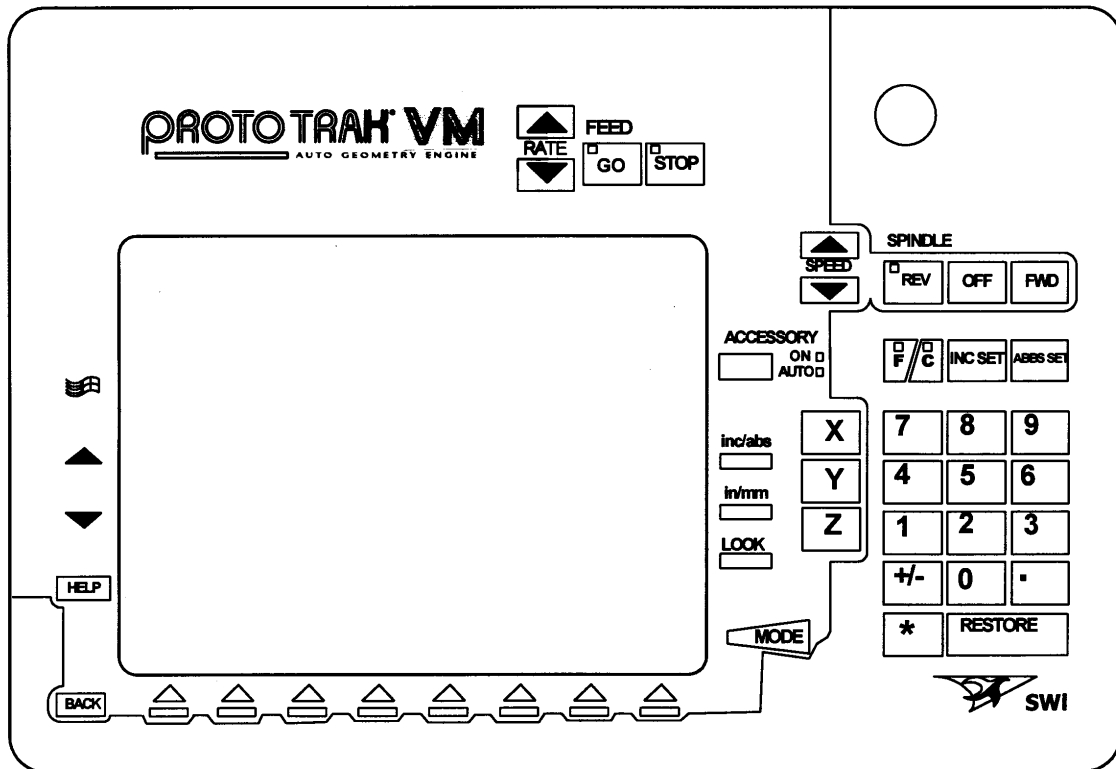


FIGURE 3.1 The ProtoTRAK VM front panel and hard keys

i00848

3.1.1 Keyboard Hard Keys

- **Feed Keys**

GO: initiates motion in Run. The green LED on the GO key will be lit when the servomotors are moving the machine either in jog or when the program run has been initiated by the GO key.

STOP: halts motion during Run. The red LED on the STOP key will be lit when the servos motors are not moving the machine.

FEED ↑: feedrate override to increase feedrate up to 150%.

FEED ↓: feedrate override to decrease feedrate down to 10%.

- **Spindle Keys**

REV: turns on spindle in reverse (counter clockwise) direction. The green LED on the REV key will be lit when the spindle is moving in reverse.

OFF: turns off spindle. The red LED on the OFF key will be lit when the spindle is off.

FWD: turns on spindle in forward (clockwise) direction. The green LED on the FWD key will be lit when the spindle is moving in the forward direction.

SPEED ↑: spindle speed override to increase spindle rpm up to 150%.

SPEED ↓: spindle speed override to decrease spindle rpm down to 10%.

ACCESSORY: Turn on accessory functions while using the machine manually in the DRO Mode, or override the programmed accessory functions by turning them on manually or turning them off. You may select to control either the coolant pump or shop air (for a spray mister) with this key. The selection of the spray mister or coolant pump control is made with a Service Code (see Section 11.4).

F/C: select between fine and course feed for the X, Y, and Z handwheels. The green LED's indicate which feed is active. Fine feed moves the axis .200 inches per revolution. Course feed moves axis .800 inches per revolution.

INC SET: loads incremental dimensions and general data

ABS SET: loads absolute dimensions and general data

INC/ABS: switches all or one axis from incremental to absolute or absolute to incremental

IN/MM: causes Inch to Metric or Metric to Inch conversion of displayed data


LOOK: part graphics in Program mode

X, Y, Z: selects axis for subsequent commands

RESTORE: clears an entry, aborts a keying procedure

0-9, +/-, . : inputs numeric data with floating point format. Data is automatically + unless +/- key is pressed. All input data is automatically rounded to the system's resolution.

MODE: to change from one mode of operation to another

 : Windows® functions including access to the Windows desktop and shut down of the ProtoTRAK VM CNC.

 : reinstates a window.

 : eliminates a window.

HELP: displays help information, math help or additional functions. Active for additional functions when the help symbol (a blue question mark) is displayed on the screen next to the HELP key.

3.1.2 Soft Keys

Beneath the display are 8 keys that are labeled with arrows. These keys are called software programmable or soft keys. A description of the function or use of each of these keys will be shown at the bottom of the display directly above each key. If, at any time, there is no description above a key, that key will not operate.

Sometimes the description or function of the key is visible but grayed out. This indicates that the particular function is not available because of some other condition. For example, if the Z retract is not set, the RUN mode key will be grayed out because setting the Z retract is a necessary step for running a program.

3.1.3 Emergency Stop Switch

The emergency stop (E-stop) switch kills all power to the spindle and ProtoTRAK's servomotors. The computer and pendant remain powered.

3.1.4 The Liquid Crystal Display (LCD)

The display of the ProtoTRAK VM is a 10.4" active-matrix color LCD. The very top is the Status Line that shows the overall status of the ProtoTRAK VM. This includes the current Mode, the current program part number, the current tool number, 2 or 3-axis mode and whether the X, Y and Z dimensions are in inch or millimeter (mm).

Just above the soft keys is a data input line that appears when an input is required.

3.2 Pendant Left Side *(See Figure 3.2 for a description of the left side panel of the display.)*

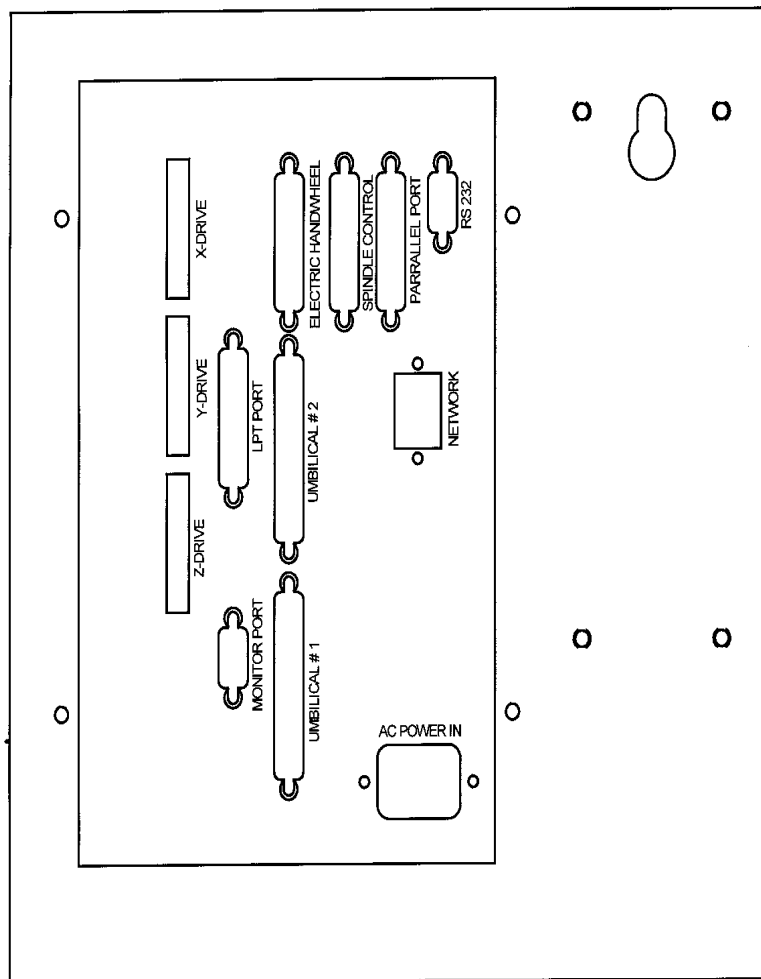


FIGURE 3.2 *The ProtoTRAK VM left side with connectors labeled*

3.3 Pendant Right Side (See Figure 3.3 for a description of the connectors and features located on the right side of the display panel.)

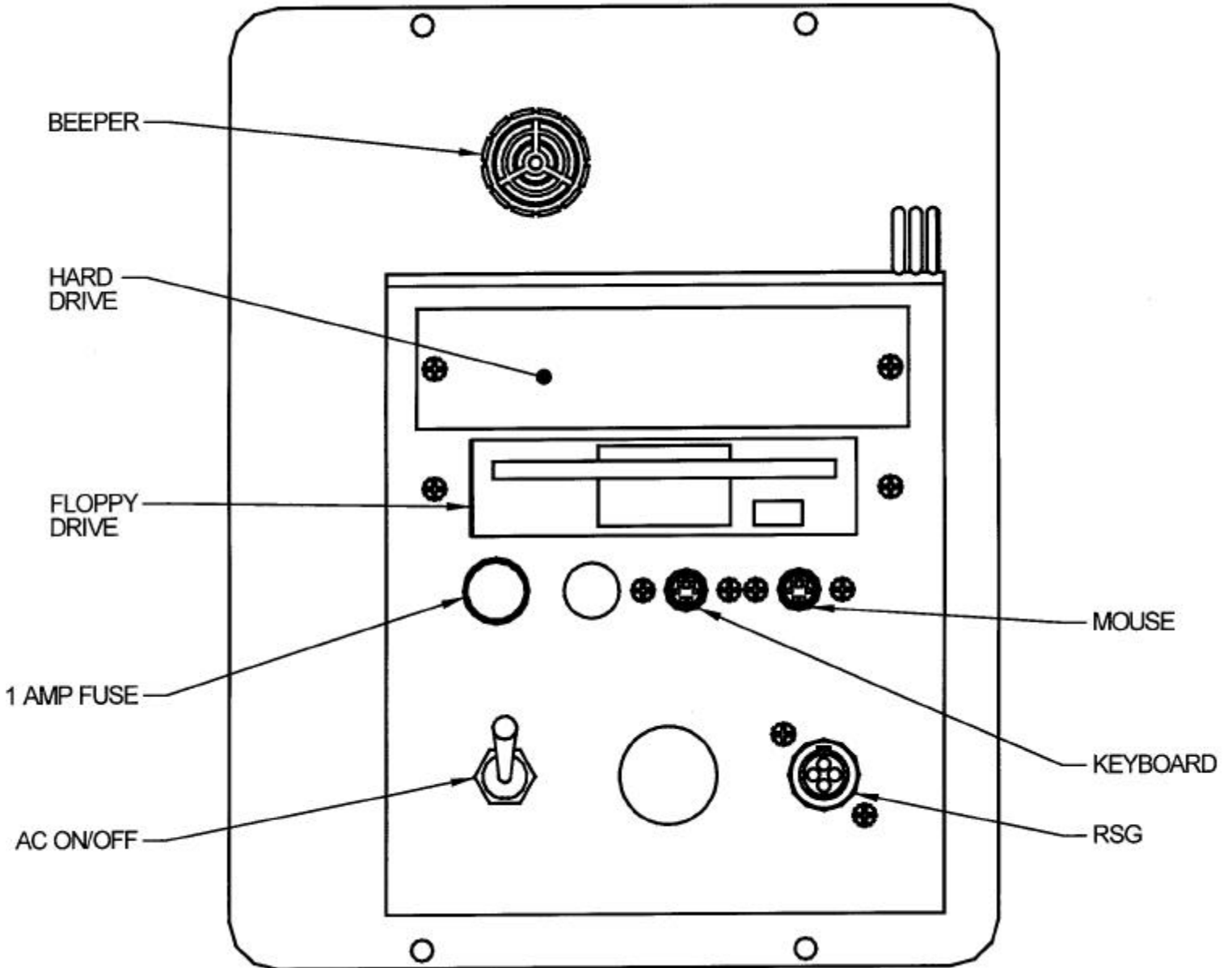


FIGURE 3.3 The ProtoTRAK VM right side

3.4 Machine Specifications (See Figures 3.4.1 and 3.4.2)

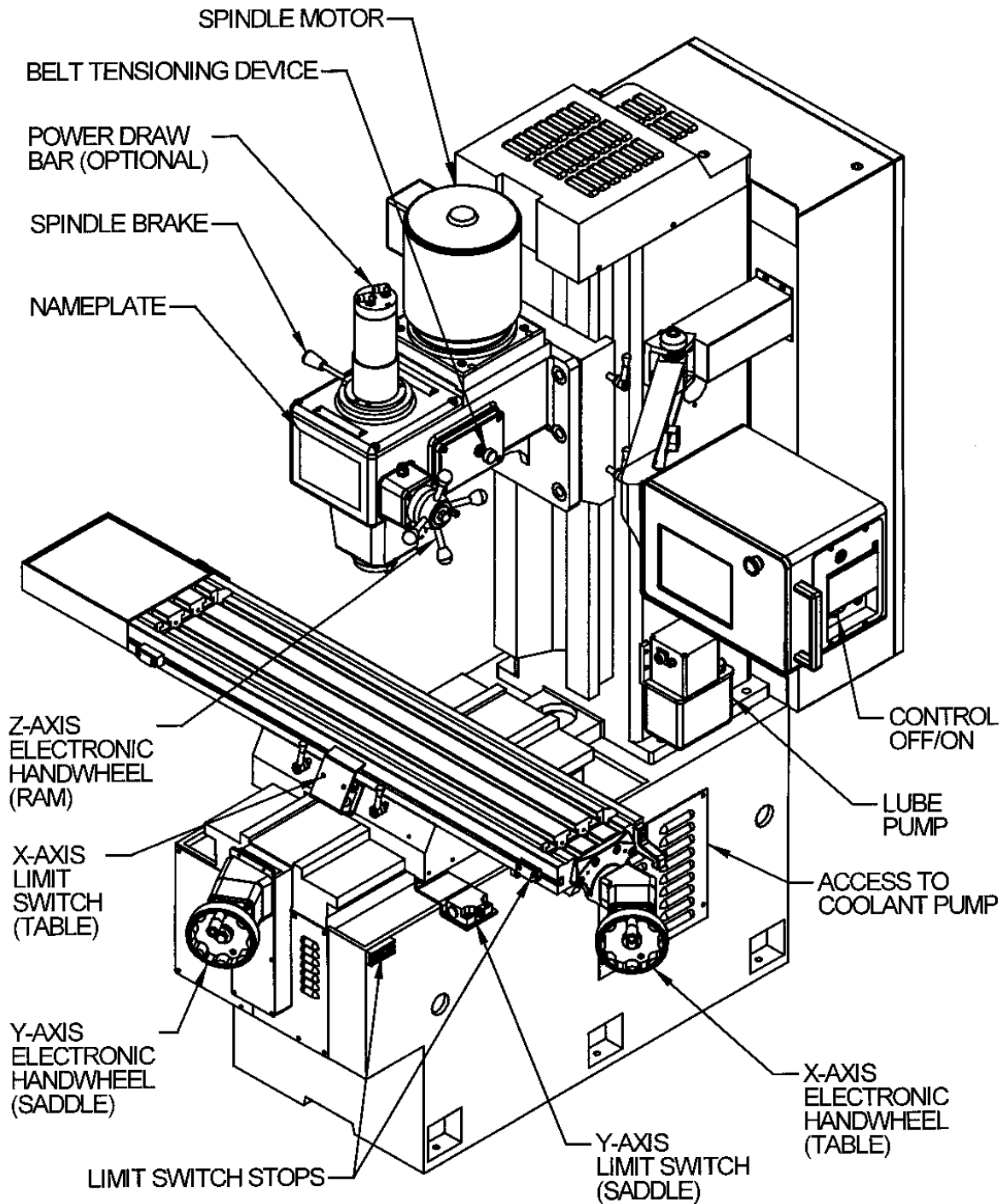


FIGURE 3.4.1 The TRAK DPMV Series machine overview

TRAK DPM V7

- Table Size - 76" x 14"
- T-Slots (No x Width x Pitch) – 3 x .63" x 2.48"
- Travel (X, Y, Z) - 60" x 23" x 20.5"
- Spindle Taper - #40 taper
- Spindle Speed Range – 200 - 5000 RPM
- Spindle Nose to Table – 0 – 21.5"
- Spindle Center to Column Face – 23"
- Spindle motor – 7.5 HP
- Power Requirement - 50 HZ/60 HZ; 3 Phase; 220v; 52 FLA
- Maximum Weight on Table - 2200 lb
- Maximum Work Capacities - Drilling (mild steel)...max. dia. 1.00"; Tapping...5/8"; Milling Capacities...7 in³/min.
- Machine Net Weight - 7650 lb.
- Shipping Weight, Dims - 7975 lb.
- Footprint (width x depth) - 37" x 62.2"
- Way Surface - Dovetail X-axis; Square Ways Y and Z-axis

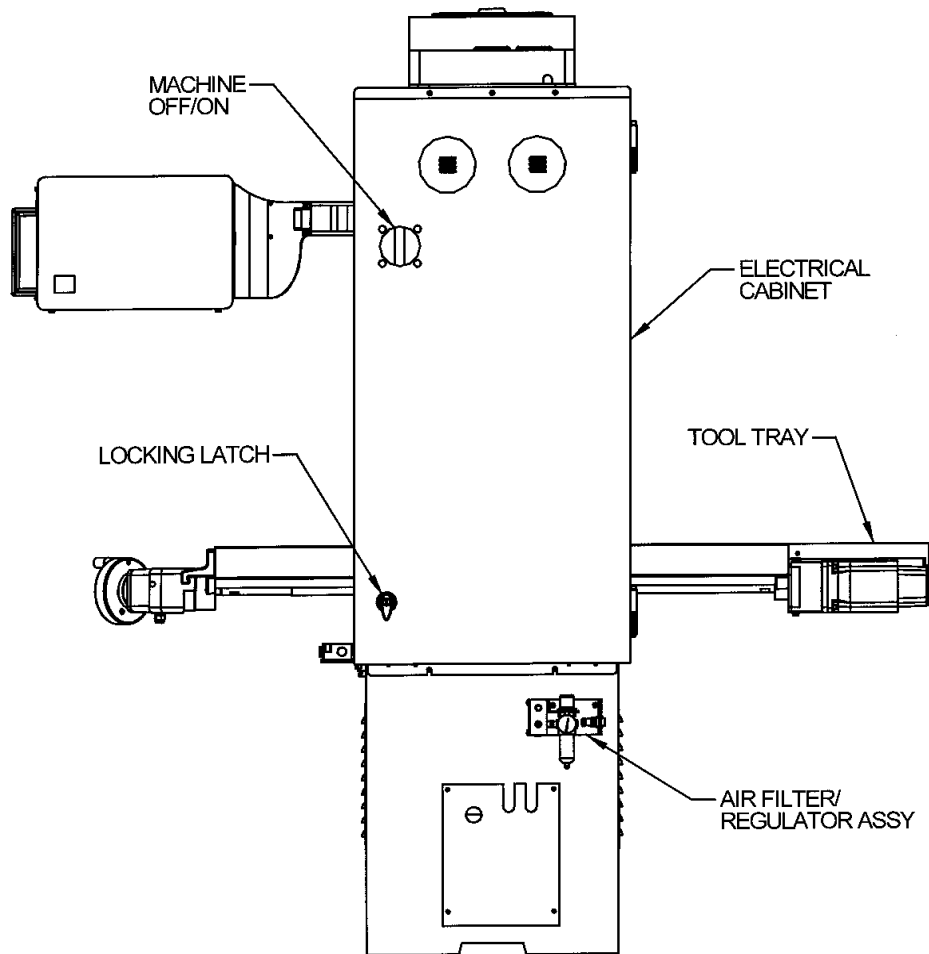


FIGURE 3.4.2 The TRAK DPMV Series back view

TRAK DPM V5

- Table Size - 50" x 12"
- T-Slots (No. x width x pitch) 3 x 5/8" x 2 9/16
- Travel (X, Y, Z) - 40" x 20" x 20.5"
- Spindle Taper - #40 taper
- Spindle Speed Range – 160 - 4000 RPM
- Spindle Nose to Table - 0 – 20.5"
- Spindle Center to Column Face – 20 ½
- Spindle motor - 5 HP
- Power Requirement - 50 HZ/60 HZ; 3 Phase; 220v; 40 FLA
- Maximum Weight on Table - 1760 lb
- Maximum Work Capacities - Drilling (mild steel)...max. dia 1.00"; Tapping...5/8"; Milling Capacities...5 in³/min.
- Machine Net Weight - 4400 lb.
- Shipping Weight - 4700 lb
- Footprint (width x depth) - 24" x 48.4"
- Way Surface - Dovetail X-axis; Square Ways Y and Z-axis

3.5 Spindle Motor

The spindle motor is a 5 HP for the DPMV5 and 7.5 HP for the DPMV7 variable speed motor that drives the spindle by a belt.

3.6 Electronic Handwheels

The electronic handwheels of the TRAK DPM V5 or V7 operate when the ProtoTRAK VM CNC is in a mode where the machinist controls the motion of the table, saddle and ram. This includes the DRO mode, the set-up mode and TRAKing in the Run mode. The electronic handwheels will not operate while the ProtoTRAK VM is moving the machine through the servomotors.

3.7 Auxiliary Functions

Auxiliary functions are controlled through the ProtoTRAK VM CNC either in the program or with the accessory key on the front panel. The Auxiliary functions consist of the following:

- Spindle off command.
- An air solenoid to control spray misters or other pneumatically activated peripheral equipment. Shop air should not exceed 125 psia.
- Switched and fused 120 VAC 8 Amp outlet(s) for coolant pumps, automatic oilers, etc.
- INPUT/OUTPUT to interface with programmable indexers, dividing heads, etc.
 - Output from ProtoTRAK VM 3 is .3-second actuation of a solid-state relay between pin 3 (plus), and pin 4 (minus).
 - Input to the ProtoTRAK VM is .3-second actuation of a solid-state relay between pin 1 (plus), and pin 2 (minus).
 - Note: Pin 1 is on top, 2 on right, 3 on left, 4 on bottom.

3.8 Limit Switches

There are limit switches for the ram, saddle and table travel.

3.9 Lubrication System

3.9.1 Factory Default Values

Interval Time – 60 min.

Discharge Time – 15 sec

Discharge Pressure – Approximately 100 – 150psi

To adjust the amount of Discharge Pressure displayed on the lube pump gauge, loosen the jam nut and turn the adjustment screw located on the top right side of the lube pump while the lube pump is activated. To activate the lube pump use Service Code "300," see Section 11.4.

CAUTION!

Failure to properly lubricate the mill will result in the premature failure of bearings and sliding surfaces.

CAUTION!

Failure to manually activate the pump at the beginning of each day, or allowing the Auto Lube to run dry may cause severe damage to the DPMV7 or DPMV5 mill way surfaces and ballscrews.

The settings for the lube pump can be viewed by doing the following: press Service Codes, press "A" (software), press Code 313. This screen lists the values programmed for the cycle time and discharge time.

3.10 Electrical Cabinet

The TRAK DPM V Series uses a single electrical source for the machine and the control. 220v power is wired directly to the disconnect switch. There are two available 110v AC outlets available for accessories.

3.11 Optional Equipment

3.11.1 Position Encoders

The ProtoTRAK VM may be configured to run either with or without independent position encoders for X and Y travel. Optional encoders include the TRAK sensors or glass scales.

3.11.2 Power Draw Bar

A manual draw bar, of the NMTB or NST type comes standard with the machine. A power draw bar option may be ordered. The draw bar included in the option may be CAT or NMTB/NST.

An NMTB/NST type of draw bar is the appropriate length to fit tool holders that have a threaded tang on the top. The CAT type is longer to thread into CNC tool holders that have the tool changer grip, or retention knob removed.

3.11.3 Remote Stop Go Switch

For the convenience of operation while running the program, a Remote Stop/Go switch may be purchased. This switch is on a ten-foot cable and operates like the FEED Stop and Go keys on the display.

3.11.4 Work Light

An optional halogen work light is available. It mounts to the left side (facing) of the column and plugs into a 110v outlet in the electrical cabinet.

3.11.5 Coolant Pump

The optional coolant pump is mounted in the back of the machine column. It is plugged into the electrical cabinet and may be configured to operate as commanded by the auxiliary functions.

3.11.6 Spray Coolant

The Fog Buster® Spray Coolant option consists of a one-gallon holding tank, nozzle, air lines and air regulator for attaching compressed air. Coolant flow is adjusted by a needle valve at the sprayer head. Air flow is adjusted at the air pressure regulator with gage. Once flows are set, sprayer operation is controlled by an air toggle switch or by interface with the Auxiliary Functions.

3.11.7 Chip Pan / Splash Shield

The Chip Pan / Splash Shield option consists of a chip pan mounted to the bed and splash shields mounted to the right and left of the column. This option is available factory-installed only. It is not available after shipment.

3.11.8 Table Guard

the table guard option provides an enclosed workspace mounted on the table. The sliding door is switched to prevent operation of the CNC Run with the door open. While it will aid in the control of chips and coolant, it is not a full, waterproof enclosure.

3.11.9 DXF File Converter

the powerful DXF file converter gives you the ability to translate CAD-generated data in ProtoTRAK programs right at your machine.

4.0 Basic Operation

One of the things that makes the TRAK DPM V5/V7 so easy to use is that most of the operations of the ProtoTRAK VM CNC are organized in Modes. Modes are logical groups of activities that naturally belong together. This eliminates the need to memorize operations – just select a mode and choose among the soft keys.

Most operations will be discussed within the section that treats the mode later in this manual. The operations described in this section either don't fit in a particular mode, or they are relevant to more than one mode.

4.1 Switching the TRAK DPM V5 or V7 On/Off

Power to the TRAK DPM V5 or V7 is turned on through the main on/off switch located on the back of the main electrical cabinet. The machine main switch must be on for power to go to the ProtoTRAK VM CNC.

4.2 Switching the ProtoTRAK VM On

To turn the ProtoTRAK VM on, move the toggle switch on the display side panel to the Up position.

The Windows operating system and the ProtoTRAK VM software will take a few seconds to load from the system's hard drive to its memory. If you have connected the ProtoTRAK VM to a network, it may take as long as 90 seconds for the communications to be established. When complete, the ProtoTRAK VM Check System screen will appear and the Windows desktop will be in the background.



FIGURE 4.2.1 This is the first screen you will see after the ProtoTRAK VM is finished booting up

The first screen encountered after power up has the F1 soft key labeled "Check System". Press this to start a routine that aligns the servomotors. Another screen will appear with a warning to remove obstacles and the F1 soft key labeled "GO". Press go and the motors will move slowly, moving the table, saddle and ram less than an inch. The process is over in 10 seconds or so.

Select the Mode of operation by pressing the soft key beneath the labeled box. Notice that the EDIT and RUN soft keys are grayed out. They will not function because there is no program in the ProtoTRAK VM. Once a program is entered, the EDIT key will function. Once a program is entered and the necessary SET-UP operations are complete, the RUN key will function.

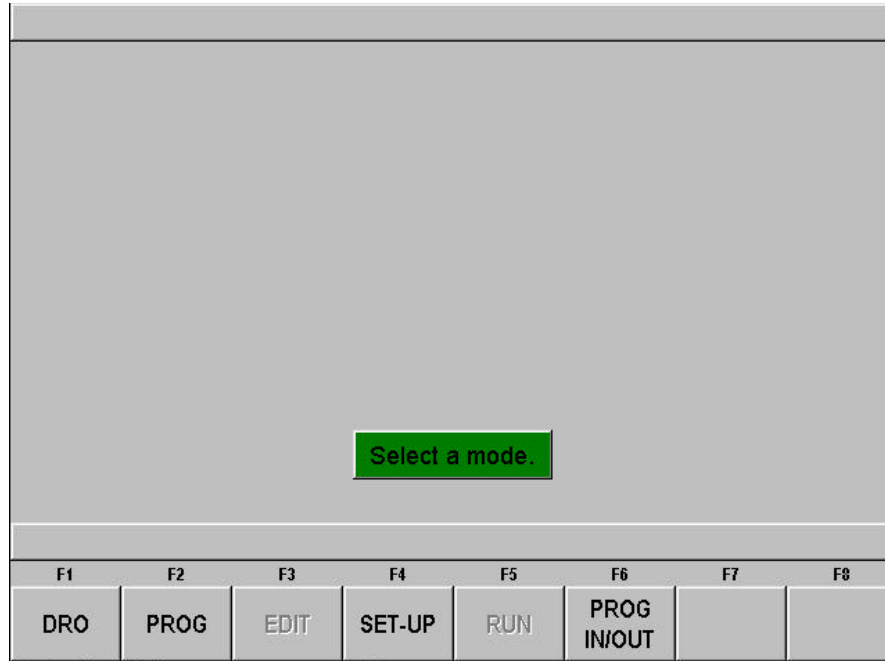


FIGURE 4.2.2 The main “select a mode” screen. Shown here, the Edit and Run Modes are grayed out because there is no program in current memory

The ProtoTRAK VM has a screen saver already programmed in. If the system is not used (either by a key stroke or by counting) for 20 continuous minutes, the display will turn itself off. The LED's on the keypad will flash every few seconds to indicate that the system is still on. Press any key or move any axis to bring the screen back to its previous display. The key you press will be ignored except to turn the screen on.

4.3 Shutting down the ProtoTRAK VM

Important: the system must be turned off properly. First press the Windows hard key and then press the SHUT DOWN soft key. After a few seconds, you will see the message "it is now safe to turn off your computer". Turn the ProtoTRAK VM off by moving the toggle switch on the display side panel to the down position.

If the CNC is not shut down properly, the system will make you wait while it runs a scan disk self-diagnostic routine and scold you for not following the instructions.

Note: When you turn the PROTOTRAK VM off, always wait a few seconds before turning it back on.

4.4 Spindle Forward/Off/Reverse

The spindle is controlled through the hard keys of the pendant. It is only operational while the ProtoTRAK VM is in the DRO or the Run modes. The spindle speed is programmed in the events during part programming (see Section 7) Programmed spindle speeds may be changed in the Edit mode (see Section 8) and the spindle speed may be set for manual operation in the DRO mode (see Section 6).

The spindle speed that is set or programmed may be overridden from 5% to 150% in 5% increments as it is running by pressing the SPEED up or down hard keys.

4.5 Manual Operation of the Ram, Table & Saddle

The TRAK DPMV5 or V7 may be used manually with the electronic handwheels. The electronic handwheels may be operated in the DRO mode and in the SET-UP mode. They have a special function in the RUN mode where they control the TRAKing feature.

The electronic handwheels will not move unless turned by you.

4.6 Emergency Stop

Press the button to shut off power to the spindle motor and axis motors. Rotate the switch to release.

4.7 Switching Between Two and Three-axis Operation

The ProtoTRAK VM may be operated as a two or three-axis CNC. Press the Windows hard key. Softkey F2 will read GO TO 2 AXIS when the ProtoTRAK VM is currently operating in three axis and it will say GO TO 3 AXIS when the ProtoTRAK VM is currently operating in two axis. See Figure 4.7.

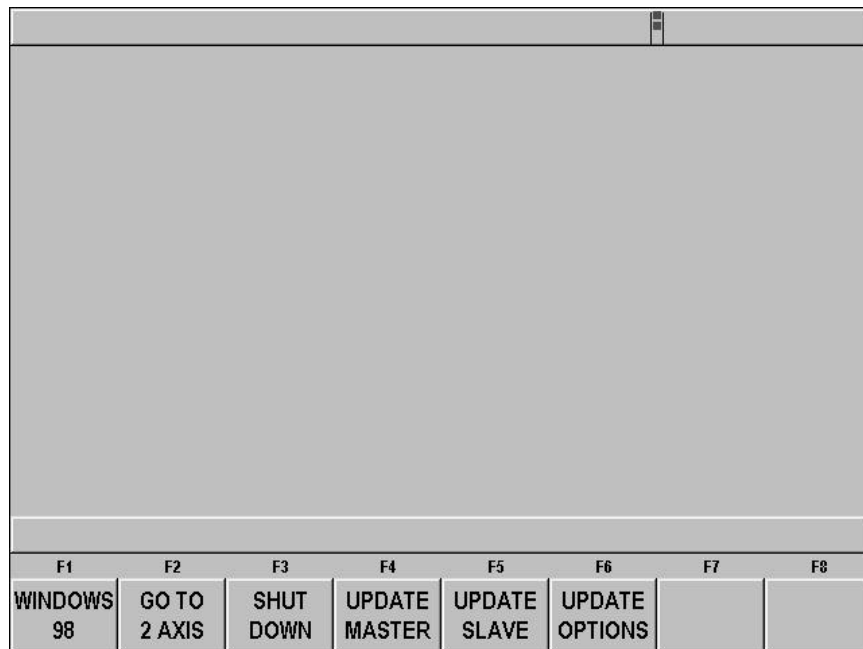


FIGURE 4.7 You will see this screen when the Windows **⓪** hard key is pressed

4.8 Spray Coolant/Coolant Pump

An optional spray coolant or coolant pump may be connected to your TRAK DPMV5 or V7. The mister or pump may be operated either manually or programmed in the events.

To operate the mister or coolant pump manually, use the ACCESSORY hard key:

- ON - will turn on the mister or coolant pump until you turn it off.
- AUTO - will turn on the mister or coolant pump when the spindle is on, as programmed and then turn it off when the spindle turns off.
- Off (no light) - the coolant pump or mister stays off.

4.9 Windows Desktop

The Windows desktop is accessed by pressing the Windows hard key and then choosing the WINDOWS 98 soft key. The ProtoTRAK VM program will continue running in the background. To get back to the ProtoTRAK VM, press the BACK hard key.

4.10 Help Functions

When a blue question mark appears next to the HELP hard key, that means special functions or configuration settings are available for the current operation. For example, at the program header with the highlight on the program name, the blue question mark appears. Pressing the HELP key at that time will call up a table with alpha and special characters you can use to name your program.

4.10.1 Math Helps

When the blue question mark does not appear, pressing HELP will initiate the Math Helps.

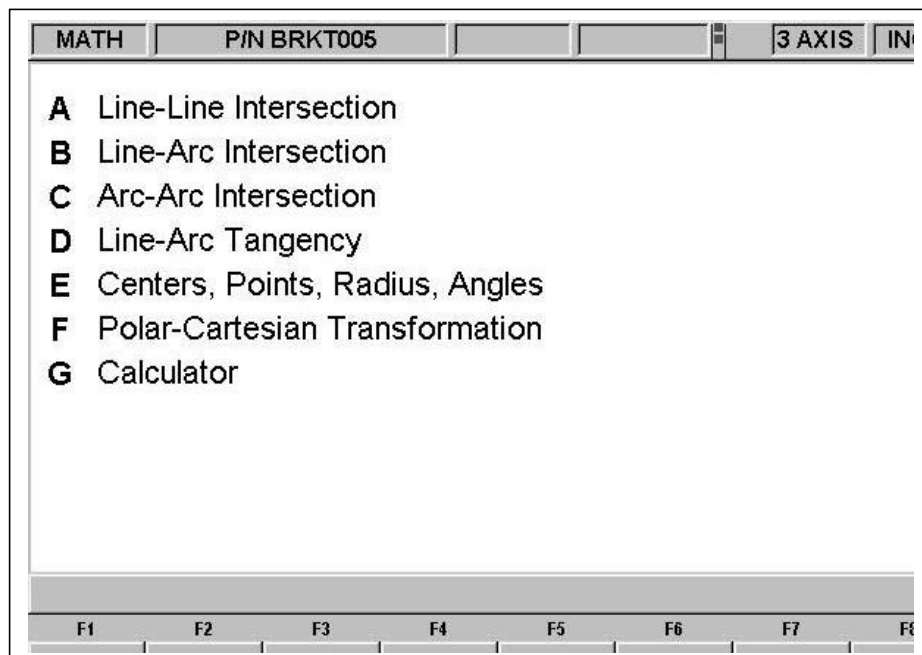


FIGURE 4.10.1 *The first Math Helps screen. Choose among the alternatives based on the information you need to calculate*

Math Helps are powerful routines that enable you to use the data you have available to calculate missing print data.

For example, Math Help type 28 enables you to solve an entire right triangle by giving two known pieces of data. To exit from the Math Help, press the Mode key.

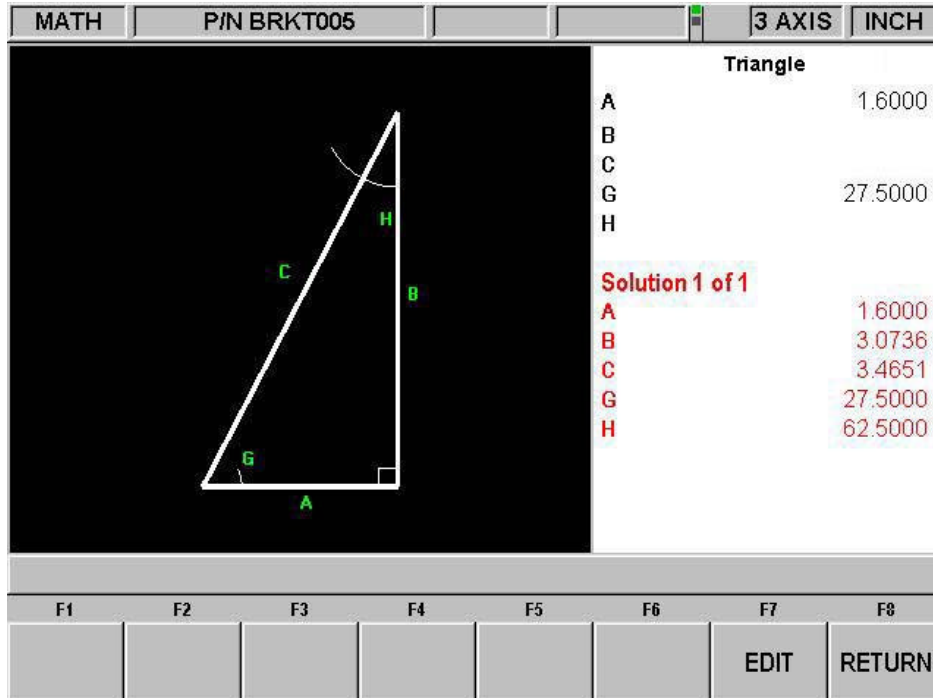


FIGURE 4.10.2 Math Help 28. In this example, by entering the length of line A and the value of angle G, the other values are calculated

You may have the Math Help solutions load directly into your program. This saves you from having to write down the solution and then key it in. While you are programming the event that needs the data from Math Help, simply press the HELP key to start the Math Help. Once a solution is obtained, you will have the following soft key selections:

Load Begin: will load the displayed solution into the event as the X and Y beginning.

Load End: will load the displayed solution into the event as the X and Y end.

Load Center: will load the displayed solution into the event as the X and Y center.

Next Solution: when there is more than one solution to the problem, this will display the alternative solutions.

Edit: this allows you to go back to the data you entered in order to make changes. Once you do this, the Resolve key will appear.

Resolve: press this to have the Math Help use the new data to give new solutions.

4.11 Windows Up or Down

Some of the selections in the ProtoTRAK VM will cause a window to appear with a message. To eliminate the window in order to see what is behind it, press the ▼ hard key. To restore the window, press the ▲ hard key.

4.12 Using a Mouse & Keyboard

A P/S 2 mouse and keyboard may be used. See Figure 2 for the location of the connectors. The ProtoTRAK VM should be turned on with the mouse plugged in so that the display will recognize the hardware.

5.0 Definitions, Terms & Concepts

5.1 ProtoTRAK VM Axis Conventions

X Axis: positive X-axis motion is defined as the table moving to the left when facing the mill. Consequently, measurement to the right is positive on the workpiece.

Y Axis: positive Y-axis motion is defined as the table moving toward you. Measurement toward the machine (away from you) is positive on the workpiece.

Z Axis: positive Z-axis motion is defined as moving the head up. Measurement up is also positive on the workpiece.

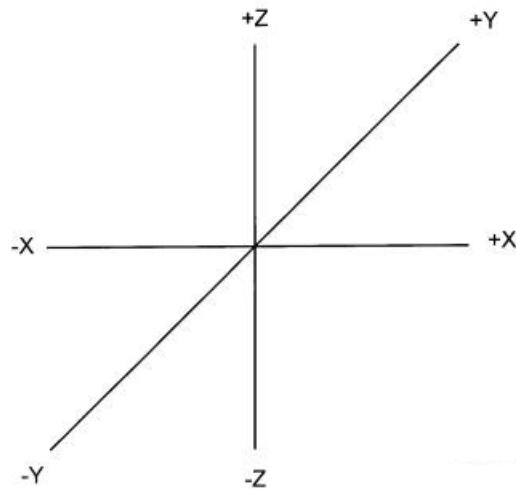


FIGURE 5.1 ProtoTRAK VM conventions

The Z RAPID dimension is the position at which Z will stop rapid traversing and switch to its programmed Z feedrate. Z motion will continue until Z End depth has been reached.

5.2 Part Geometry & Tool Path Programming

The ProtoTRAK VM gives you ultimate flexibility in programming. Programs that are entered through the ProtoTRAK VM system can be entered as either Part Geometry or Tool Path.

Part Geometry programming is the popular programming style of the ProtoTRAK family of products. This is done by defining the final geometry of the part, and the ProtoTRAK VM has the job of figuring out the tool path from the part dimensions and the tool set-up information. This is a great benefit compared to regular CNC because it doesn't force the programmer to do the difficult job of defining tool path. A consequence of part geometry programming is that the following are not allowed:

- connection of an incline plane and another event
- connection of two events that lie in different vertical planes

Using Geometry Programming, it is impossible for the ProtoTRAK VM to calculate a tool path for these cases without creating a problem. In cutting the geometry desired in the first event, the tool ends up out of position for the next event. Resolving the difference in tool position where the first event ends and the next event begins means either the CNC

calculates and makes an unprogrammed move, or it retracts the tool out and then back into the part.

These cases are not encountered often, but when they are you have the option of using Tool Path programming. In Tool Path programming you define the events the same way, but all inputs are treated as tool center. It is your job to calculate and program the tool path.

5.3 Planes and Vertical Planes

A plane is any flat surface. If that surface lies flat on the table, it is the XY plane. That is, if you move your finger along that surface or plane, you are moving in the X and/or Y direction, but not in Z (or at least not until you pick your finger up). If you tilted that surface (think of it as a piece of paper) straight up so that it faces the front of the machine, it would be in the XZ plane. If you tilted it up so that it faced left or right, it would be in the YZ plane.

A vertical plane is any plane (or surface) tipped up on its edge on the table (see below).

Unlike most CNC controls, the ProtoTRAK VM can machine arcs in any vertical plane rather than just XZ or YZ.

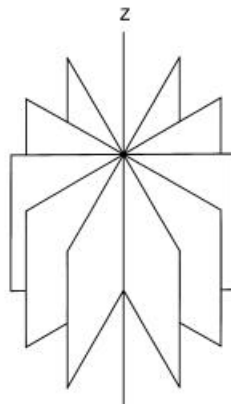


FIGURE 5.3 Vertical planes

5.4 Absolute & Incremental Reference

The ProtoTRAK VM may be programmed and operated in either (or in a combination) of absolute or incremental dimensions. An absolute reference from which all absolute dimensions are measured (in DRO and program operation) can be set at any point on or even off the workpiece.

To help understand the difference between absolute and incremental position, consider the following example:

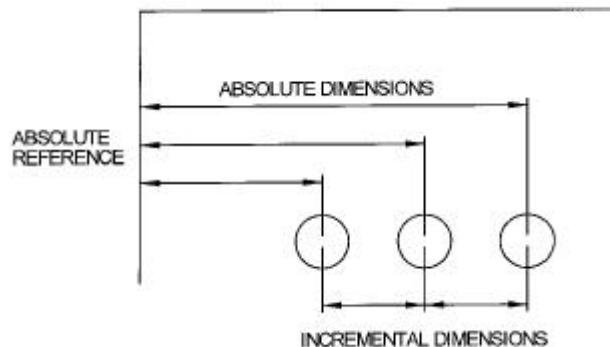


FIGURE 5.4 Each point has both an absolute and an incremental reference in the X axis. The ProtoTRAK VM allows you to program using either.

5.5 Referenced & Non-Referenced Data

Data is always loaded into the ProtoTRAK VM by using the INC SET or ABS SET key. X, Y, Z positions are referenced data. In entering any X, Y, or Z position data, you must note whether it is an incremental or absolute dimension and enter it accordingly. All other information (non-referenced data), such as tool diameter, feedrate, etc. is not a position and may, therefore, be loaded with either the INC SET or ABS SET key. This manual uses the term SET when either INC SET or ABS SET may be used interchangeably.

5.6 Incremental Reference Position in Programming

When X, Y, Z RAPID and Z data for the beginning position of any event are input as incremental data, this increment must be measured from some known point in the previous event. Following are the positions for each event type from which the incremental moves are made in the subsequent event:

Position: X, Y and Z programmed

Drill: X, Y, Z RAPID, and Z END programmed

Bolt Hole: X CENTER, Y CENTER, Z RAPID and Z END programmed

Mill: X END, Y END, Z RAPID and Z END programmed

Arc: X END, Y END, Z RAPID and Z END programmed

Circle (POCKET or FRAME): X CENTER, Y CENTER, Z RAPID and Z END programmed

Rectangle or Irregular (POCKET or PROFILE): X1 and Y1 corner, Z RAPID and Z END programmed

Helix: The X END, Y END, Z RAPID, and Z END programmed

Sub: The reference position as defined for the specific events above for the event prior to the first event that was repeated.

A.G.E. PROFILE: The appropriate reference position as defined for the specific events above for the last event that is programmed

For example, if an ARC event followed a MILL event, a 2.0 inch incremental X BEG would mean that in the X direction the beginning of the ARC event is 2.0 inches from the end of the MILL event.

5.7 Tool Diameter Compensation

Tool diameter compensation allows the machined edges shown directly on the print to be programmed instead of the center of the tool. The ProtoTRAK VM then automatically compensates for the programmed geometry so that the desired results are obtained.

Tool cutter compensation is always specified as the tool either right or left of the workpiece while looking in the direction of the tool motion.

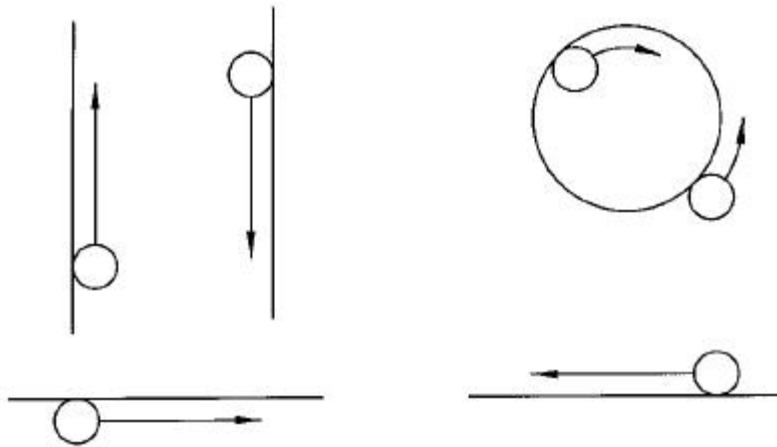


FIGURE 5.7.1 Examples of tool right

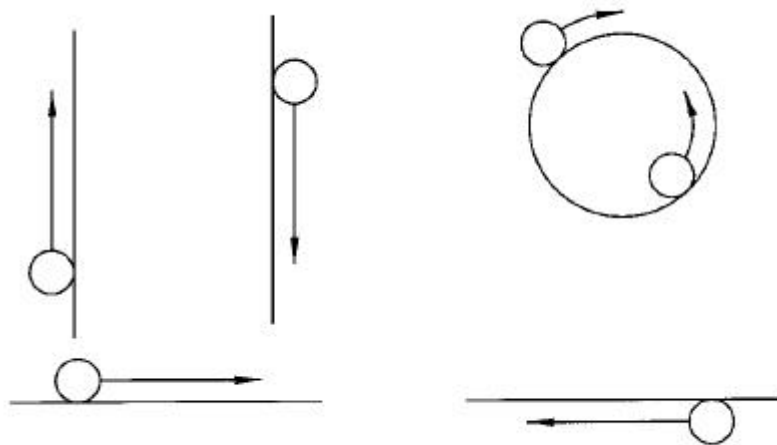


FIGURE 5.7.2 Examples of tool left

Tool center means no compensation either right or left. That is, the centerline of the tool will be moved to the programmed points.

5.8 Tool Diameter Compensation when Contouring in Z with Part Geometry

Left and right tool diameter offsets are always those projected into the XY plane. Tool offsets in the Z direction are always up and assume the use of a ball end mill. When contouring in the Z-axis, this up tool offset is always activated regardless of left, right, center if the Part Geometry option is selected. There is no Z-axis up tool offset applied when the Tool Path option is selected.

Special attention must always be paid to tool offsets when machining with a ball end mill. The reason for this is that the tool diameter changes in the bottom part (that portion equal to the tool radius) of the tool.

The tool is always positioned at the beginning of a milling operation so that the correct point on the ball end of the tool is tangent to the beginning point, and offset perpendicular to the machined edge by the radius of the tool. Consider the example below of milling a ramp in the XZ plane from point B to point C.

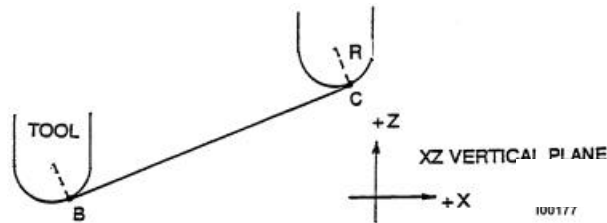


FIGURE 5.8.1 Ball end mill position with respect to program points. Tool starts so end mill is tangent to BC. R from center of tool is perpendicular to BC

Note how the tool at the beginning point (point B) starts below (in the Z direction) point B so that it can actually touch this point. If this were not true, a cusp would remain to the left of point B.

Now consider a similar example milling from A to B to C in the XZ plane.

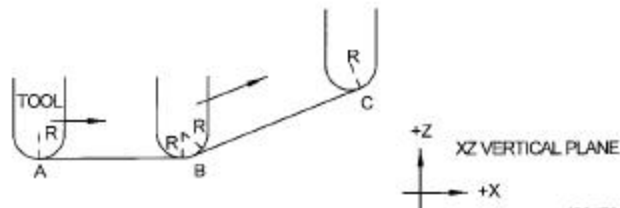


FIGURE 5.8.2 In order to respect the lines defined by the programmed points, the ball end mill never touches point B. Tool starts centered over A offset up by the tool radius R. It moves right until it is tangent to both AB and BC. Then moves to point C as in the first example

Note the Tool at B does not drop below the AB line and, therefore, never touches point B. As a result, a fillet is formed at point B equal to the tool radius.

This second example of continuous machining from one cut (AB) to another (BC) with full cutter compensation between requires the two cuts to be made with events which are connective (see Section 5.9 or 5.10 for a more complete discussion of this requirement).

5.9 Connective Events

Connective events occur between two milling events (either Mill or Arc) when the X, Y, and Z ending points of the first event are in the same location as the X, Y, and Z starting points of the next event. In addition, the tool offset and tool number of both events must be the same. And both events must lie in the XY plane or the same vertical plane (see Section 3.2).

5.10 Conrad

Conrad is a unique feature of the ProtoTRAK VM that allows you to program a tangentially connecting radius between connective events, or tangentially connecting radii for the corners of pockets and frames without the necessity of complex calculations.

For the figure below, you program an Arc event from X1, Y1 to X2, Y2 with tool offset left, and another Arc event from X2, Y2 to X3, Y3 also with tool offset left. During the programming of the first Arc event, the system will prompt for Conrad at which time you input the numerical value of the tangentially connecting radius $r=k3$. The system will calculate the tangent points T1 and T2 and direct the tool cutter to move continuously from X1, Y1 through T1, $r=k3$, T2 to X3, Y3.

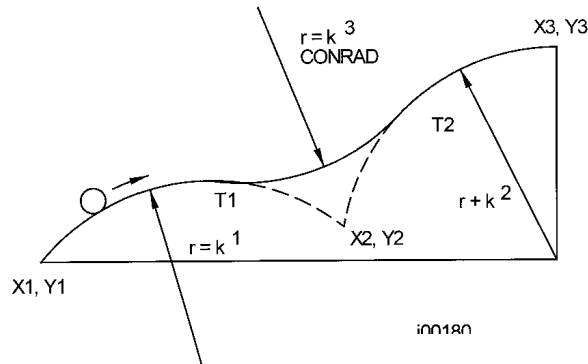


FIGURE 5.10.1 A Conrad is added between the two intersecting lines

Note: Conrad must always be the same as or larger than the tool radius for inside corners. If Conrad is less than the tool radius, and an inside corner is machined, the ProtoTRAK VM will ignore the Conrad.

For the figure below, you program an Arc event from X1, Z1 to X2, Z2, and a Mill to X3, Z3. During the programming of the Arc event, the system will prompt for Conrad at which time you input the numerical value of the tangentially connecting radius $r=k$. The system will calculate the tangent points T1 and T2 and direct the tool cutter to move continuously from X1, Z1 through T1, $r=k$, T2 and on to X3, Z3.

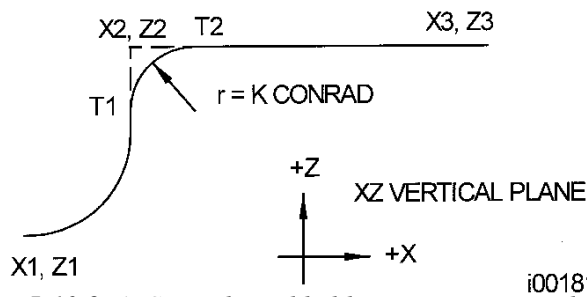


FIGURE 5.10.2 A Conrad is added between an arc and a line

5.11 Memory & Storage

Computers can hold information in two ways. Information can be in **current memory** or in **storage**. Current memory (also known as RAM) is where the ProtoTRAK VM holds the operating system and any part program that is ready to run. While a program is being written, it is in current memory. Storage of information is on the ProtoTRAK VM hard drive, a floppy disk or an offline computer connected to the ProtoTRAK VM via the RS232 or network connection.

6.0 DRO MODE

The ProtoTRAK VM operates in DRO Mode as a sophisticated 3-axis digital readout with jog and power feed capability.

6.1 Enter DRO Mode

Press **MODE**, select **DRO** soft key. The screen will show:

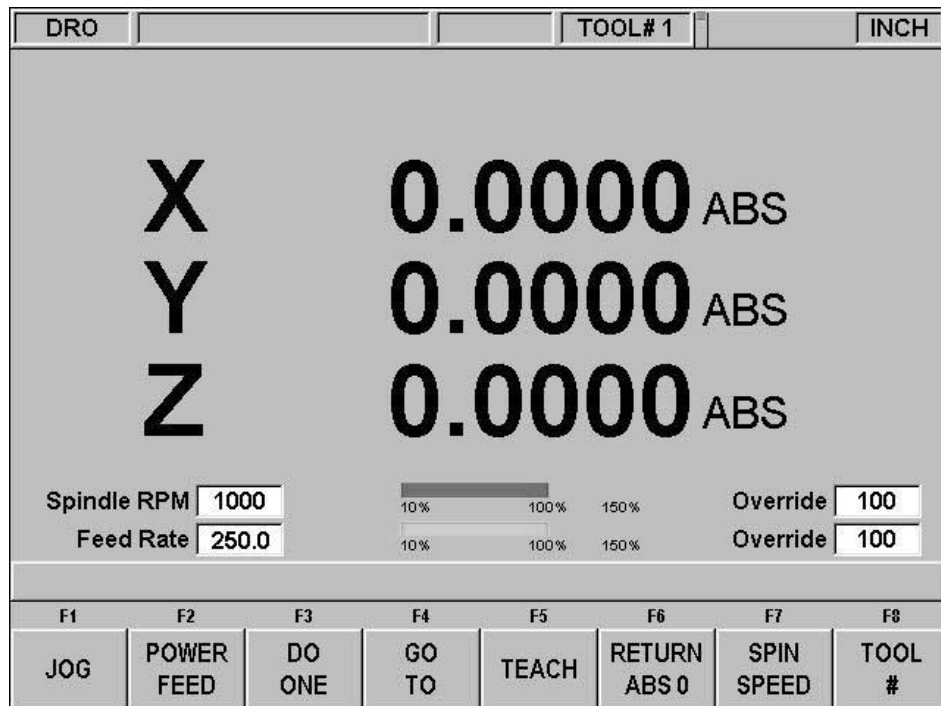


FIGURE 6.1 The DRO screen

Note the **RETURN** soft key is lit when in Jog or Power Feed operation.

6.2 DRO Functions

Clear Entry: Press RESTORE, then re-enter all keys.

Inch to MM or MM to Inch: Press IN/MM and note LCD screen status line.

Reset One Axis: Press X or Y or Z, INC SET. This zeros the incremental position in the selected axis.

Preset: Press X or Y or Z, numeric data, INC SET to preset selected axis.

Reset Absolute Reference: Press X or Y or Z, ABS SET to set selected axis absolute to zero at the current position.

Note: This will also reset the incremental dimension if the absolute position is being displayed when it is reset.

Preset Absolute Reference: Press X or Y or Z, numeric data, ABS SET to set the selected axis absolute to a preset location for the current machine position.

Note: This will also reset the incremental dimension if the absolute position is being displayed when it is preset.

Recall Absolute Position of All Axes: Press INC/ABS. Note the dimension for each axis is labeled INC or ABS. Press INC/ABS again to revert to the original reading.

Recall Absolute Position of One Axis: Press X or Y or Z, INC/ABS. Note the INC or ABS label for each axis. Repeat to get selected axis back to original reading



6.3 Jog

The servomotors can be used to jog the table, saddle and ram.

- a. Press the **JOG** soft key.
- b. A flashing message will appear saying "CAUTION: JOG KEYS ARE ACTIVE".
- c. To jog, press the X, Y or Z hard keys.
- d. To stop jogging, release the key.
- e. The speed of jog is displayed in the box next to the words "Feed Rate" on the lower left side of the LCD screen.
- f. Press the +/- hard key to reverse direction. When the number in the Feed rate box is negative, this indicates the minus direction.
- d. Press the RATE keys to reduce and to increase the jog speed in 10 percent increments. The changes in speed may be seen in the Feed rate box and on the green feed rate indicator. The amount of override is displayed in the Override box.
- g. To jog at a certain rate, simply enter that number as inches or mm per minute and then press the X, Y or Z key. You may also use the override key to adjust this number. Press **RSTR** to return to 250 ipm or 6350mm.
- h. Press **RETURN** soft key to return to manual DRO operation.

6.4 Power Feed

The servomotors can be used as a power feed for the table, saddle or quill, or all three simultaneously.

- a. Press the **POWER FEED** soft key.
- b. A message box will appear that shows the power feed dimensions. All power feed moves are entered as incremental moves from the current position to the next position.
- c. Enter a position by pressing the axis key, the distance to go and the +/- key (if needed). Input the entry by pressing INC SET. For example, if you wanted to make a power feed move of 2.00" of the table in the negative direction, you would enter: X, 2, +/-, INC SET.
- d. Initiate the power feed move by pressing GO.
- e. The feedrate is automatically set to 10 ipm (or 254 mm per min). Press **FEED**  or **FEED**  to adjust the feedrate from 1 ipm to 100 ipm. (or 254 to 2540).
- f. Press **STOP** to halt power feed. Press **GO** to resume.

- g. Repeat the process beginning at "c" above as often as you wish.
- h. Press **RETURN** soft key to return to manual DRO operation.

6.5 Do One

The Do One routines in the DRO mode allow you to do one CNC operation while machining manually without having to write a program.

The programming and tool path of the events in Do One are nearly identical to those in the Program Mode. See Section 8 for instructions for programming.

6.6 Go To

The Go To function in the DRO mode allows you to set a dimension in X, Y or Z at which you want the machine to stop moving when you are cranking manually. For example, if you wanted to machine manually exactly 2" of table motion, you would input: Go To, X, 2, Inc Set. While the Go To window is displayed, the ProtoTRAK VM will not let you pass that 2" dimension you set.

- a. Press the Go To key.
- b. Enter the axis, X, Y, Z or any combination.
- c. Press Inc Set or Abs Set.
- d. Crank the handwheel. Motion will stop at the entered dimension even if you continue to crank the handwheel.

6.7 Teach

Teach gives you the ability to enter X and Y dimensions into a program. It can be a useful way of entering a few manual moves for operations like clearing out excess material or remembering a few hole locations.

The process of using Teach is in two parts. The first part takes place in the DRO Mode. This is where you start the Teach program, establish the program events and enter the X and Y dimensions. The second part is in the Program Mode. This is where you complete the Teach events that you began in the DRO Mode by entering the rest of the data. Once the data is entered, the Teach events become just like the other events that make up a program.

6.7.1 Entering Teach Data

From the DRO screen, press Teach.

On the top of the screen, you will see the message "Teach" and an event counter. When you enter Teach, you are actually programming events. If there is already a program in current memory, Teaching will add events to the end of the program. If there is not already a program in current memory, Teaching will start a new program. For example, if you already had a program in current memory that had 10 events, when you press Teach, the event counter will say EVENT 11. If there was no program, the event counter will say EVENT 1.

The event counter shows the event for which data is being entered. You may teach in position, drill and mill events only.

On the first Teach screen, the softkeys are:

POSN: a position move. For two-axis programming, the POSN and DRILL events are combined.

DRILL: a drill or bore.

MILL BEGIN: the beginning of a straight line or MILL event.

END TEACH: ends the teaching process and returns you to the main DRO screen.

If you press the POSN or DRILL key, the event counter will go up by one and the screen remains the same. If you press the MILL BEGIN key, the event counter stays on the same number. That is because you have given the beginning point of the line but not yet the end. The softkey selections will change to:

MILL END: the last point of the Mill event. Press this to end the Mill event and select a POSN, DRILL or new MILL event.

MILL CONT: the last point of the current Mill event, but the beginning of the next Mill event. You may enter successive Mill events by pressing the MILL CONT key.

Pressing either of the above softkeys will cause the event counter to increase by one.

At any time you may exit the Teach and return to the DRO screen. The events you have defined with their X and Y dimensions are finished in the Program Mode. See Section 8.15.

6.8 Return to Absolute Zero

At any time during manual DRO operation you may automatically move the table to your absolute zero location in X and Y by pressing the **RETURN ABS 0** soft key. When you do, the message window will read "Ready to Begin: Press Go when Ready". Make sure your tool is clear and press the **GO** key. The servos will turn on, move Z to Z retract then move the table at rapid speed to your X and Y absolute zero position, and then turn off. You will be at zero and in manual DRO operation. When you are in 2-axis CNC operation, only the X and Y will move, the head will not.

6.9 Spindle

Spindle operation is controlled through the ProtoTRAK VM when machining manually in the DRO mode.

6.9.1 Spindle Speed

Press the SPIN SPEED soft key. The Data Input Line will prompt "Spindle RPM". Enter the RPM value from 200-5000 for the DPMV7, and 160-4000 for the DPMV5

Enter this spindle speed by pressing SET.

If the spindle was already on, it will continue to spin at the RPM set previously until the new speed is entered into the ProtoTRAK VM by pressing SET.

6.9.2 Spindle Hard Keys

In the DRO mode, the spindle hard keys on the display are active.

6.10 Tool

The ProtoTRAK VM allows you to use the offsets for tools in your Tool Table (see Section 11.1) in the DRO Mode. To change tools, press the TOOL # soft key and enter the tool number when prompted by the Data Input Line.

When you set up a tool in the Set-Up Mode, if you do not wish to use the tools in the Tool Table, simply ignore the Tool # feature.

7.0 Program Mode

Part 1: Getting Started & Some General Information

7.1 Programming Overview

The ProtoTRAK VM makes programming easy by allowing you to program the actual part geometry as defined by the print.

The basic strategy is to fill in the initial program information and then program the features of the part by selecting the soft key event types (geometry) and then follow all instructions in the Data Input Line.

When an event is selected, all the prompts that need to be input will be shown on the right side of the screen. The first prompt will be highlighted and also shown in the Data Input Line. Input the dimension or data requested and press **INC SET** or **ABS SET**. For X, Y or Z dimension data it is very important to properly select **INC SET** or **ABS SET**. For all other data either **SET** will do.

As data is being entered it will show in the Data Input Line. When **SET**, the data will be transferred to the list of prompts in the right side of the screen, and the next prompt will be shown in the Data Input Line.

When all data for an event has been entered, the entire event will be shifted to the left side of the screen and the conversation line will ask you to select the next event.

7.2 Enter Program Mode

Press **MODE**, select **PROGRAM** soft key.

The ProtoTRAK VM will allow only one program in current memory. To write a new program, you must first erase the one in current memory (you may want to first store the program for use in the future). If there is already a program in current memory, entering the Program mode will allow you to edit or add to that program.

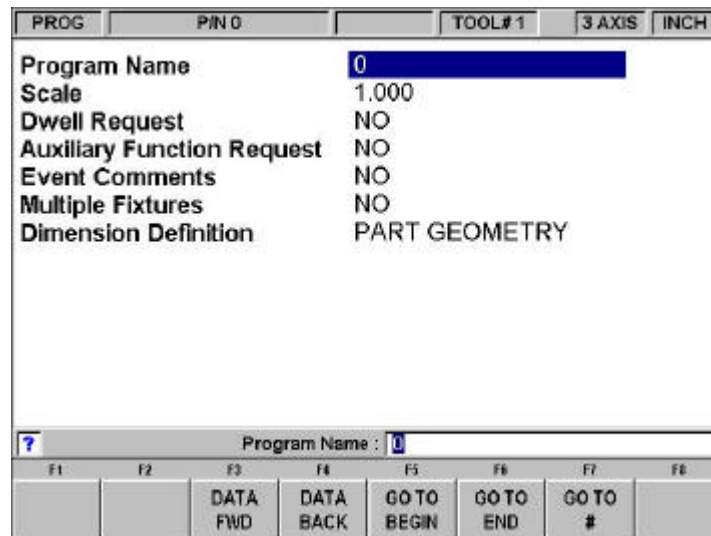


FIGURE 7.2 The Program Mode header screen

7.3 Program Header Screen

The first screen you see when you enter the Program Mode is the Program Header Screen. The Program Header Screen gives you options that apply to the entire program. The softkey selections allow you to enter the program at any point.

The program name and general programming options you choose in the Program Header Screen will be summarized in the program as "Event 0".

7.3.1 Program Name

Programs written on the ProtoTRAK VM are usually named for the part that is to be machined. When programs (or files) are named using the ProtoTRAK VM, the name can be up to 20 characters long. Programs imported into the ProtoTRAK VM may be longer. While 20 characters are allowed, the entire program name may not be shown in the status line or the program header screen.

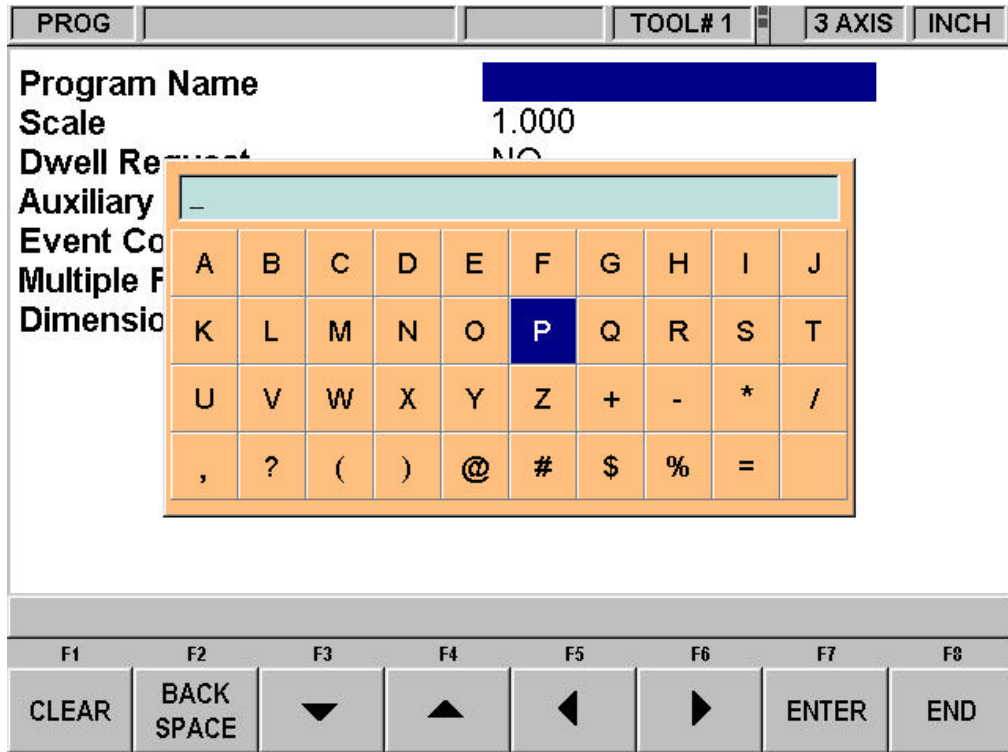


FIGURE 7.3 Pressing the Help hard key when the Program Name is highlighted calls up alpha keys

Program names can include numbers, letters, spaces and other characters. When the Program name prompt is highlighted, the Data Input Line will show "Program Name:". At this point you may:

- Press number keys.
- Press Help to access alpha keys and special characters in the ProtoTRAK VM.
- Use a keyboard plugged into the ProtoTRAK VM to name the program.

To use the alpha keys and special characters on the ProtoTRAK VM:
Use the Clear softkey to erase the entire line; the Backspace softkey to erase the last character or number.

- Use the arrow softkeys to move around the table.
- Once the character you want is highlighted, use the Enter softkey to load the character into the program name.
- Use the blank space on the lower right of the table to insert a space into the program name.
- Once you finish entering the letters and special characters, press the End softkey. This tells the ProtoTRAK VM that you are finished with the alpha table. Numbers may still be added to the program name.

When you are finished with the program name, press SET to enter it into the current memory.

Note: It is not necessary to enter a part number. If none is entered and a GO TO soft key is pushed, the system will assume a part number 0.

7.3.2 General Program Options

Use the DATA FWD softkey to select general programming options.

Scale: Allows a scale factor between .1 and 10. An input of 5 means the part will be 5 times as big as the programmed dimensions. A value of 1.0000 is assumed if nothing is input.

Dwell Request: Allows you to input a dwell at the bottom of a drill bolt hole or bore cycle for events you select. Select the appropriate **YES** or **NO** soft key. If you select **YES** you will be prompted to input a dwell time in seconds from .1 to 99.9 when appropriate to the event being programmed.

Auxiliary Function Request: Asks if you wish to activate any of the optional auxiliary functions (see Section 7.4) at any time during the program. Select the appropriate **YES** or **NO** soft key. If you select **YES** you will be prompted to input the type and sequencing of the auxiliary functions during event programming.

Event Comments: If you select "Yes" for event comments, you will have the opportunity to insert a comment in each event. For Irregular Pocket and Irregular Profile events, you will be able to enter a comment at the header event, but not for each A.G.E. Turn and A.G.E. Arc event.

Comments you insert will appear in the Data Input Line as the event begins to run. Comments may be composed of letters, numbers and some symbols and may be up to 20 characters.

While programming the event with the Event Comments set to Yes, when the highlight is on the Event Comments prompt, you may enter a comment using the same methods used to enter a program name, as described above.

Multiple Fixtures: Asks you if you wish to turn on the multiple fixtures offset. Answering Yes will cause a prompt to appear at each event asking which fixture the event was referenced from. If you select Yes, the Data Input Line will ask you to enter a fixture default number from one to six. The fixture default number is the fixture that will be applied to all the events in current memory when Multiple Fixtures is turned on or

when a new event is programmed without another event being specified. Enter the default fixture, or leave the number unchanged, and press SET. Multiple Fixtures are explained more fully in Section 7.5.

Dimension Definition: The ProtoTRAK VM gives you a choice in programming either tool path or geometry. Part Geometry programming allows you to define the geometry you want your part to have and then the CNC does the more difficult job of calculating tool path for you automatically. This is a great benefit for most parts most of the time because it means that the CNC is doing the hard work of determining tool position.

One restriction to part geometry programming is that for events to be connective, they must lay on the same vertical plane (see Section 5.3 for a definition of vertical planes). For this reason, the ProtoTRAK VM gives you the option of entering your own tool path. If you wish to program the part by defining tool path yourself, you may choose the TOOL PATH softkey. Otherwise, Part Geometry programming is assumed. Tool Path operates under the same rules as Haas standard RS274.

A program must be entirely written in Part Geometry or Tool Path programming.

7.3.3 Program Header Softkeys

The following softkeys are encountered in the Program Header Screen. The first five listed below are always there. The last four appear when relevant to the general programming option.

DATA FWD: moves the highlight forward through the programming options without setting an input into the program.

DATA BACK: moves the highlight backward through the programming options without setting an input into the program.

GO TO BEGIN: puts the Program Header on the left side of the screen and the first event on the right side.

GO TO END: puts the last programmed event on the left side of the screen and the next event to be programmed on the right side.

GO TO #: enter the event number you wish to go to and then press SET. Puts the requested event number on the right side of the screen and the previous event number on the left.

Note: for a new program that has no Events, all the GO TO selections will take you to the beginning, with the program header information summarized on the left (as Event 0) and the Select an Event options for Event 1 on the right.

YES and NO: Yes and no appear when the Dwell Request, Auxiliary Function Request and the Event Comments are highlighted. Choosing Yes will give you prompts for using these options while you are programming. You may return to the Program Header Screen at any time to choose or cancel these prompts.

PART GEO: sets up the programming as Part Geometry.

TOOL PATH: sets up the programming as Tool Path.

7.4 Auxiliary (AUX) Functions

The ProtoTRAK VM can control four different auxiliary functions. You can select whether to activate or deactivate these functions at the beginning or end of each event.

If Auxiliary Functions are selected on the program header, the system will prompt for AUX BEG and AUX END in each event.

When running programs with Auxiliary functions, the ACCESSORY hard key on the front panel must be in the correct position. If you want the program to automatically turn the Auxiliary functions on and off, press the ACCESSORY key until the light is on in the AUTO position.

AUX BEG options:

Input:	Function	Comments
0	None	No Auxiliary functions will begin when this event begins to run.
1	Coolant	The coolant pump will be turned on when this event begins to run.
2	Air on	The air will turn on (usually for a spray mister) at the beginning of this event.
3	Pulse indexer	Activates a 0.3 second electronic pulse at the beginning of the event. See note below.

AUX END options:

0	None	No Auxiliary functions will turn off at the end of this event.
1	Coolant off	Turns the coolant off at the end of this event.
2	Air off	Turns the Air off at the end of this event.
3	Pulse Indexer	Activates a 0.3 second electronic pulse at the end of this event. See note below.
4	Spindle	Turns off the spindle at the end of this event. Note, the spindle automatically turns off for each tool change – it is not necessary to program a spindle off.

Coolant or air on and off is automatically programmed for tool changes. If you want the spray mister or coolant pump on while cutting the entire part, you need only program the Aux begin on the first event and the Aux end for the last event. The spray mister or coolant will turn on when the spindle turns on and will turn off during tool changes.

The Pulse Indexer function is designed to operate with a standard indexer. Programming an Aux 3 at the end of an event will cause the ProtoTRAK VM to stop machining at the end of the event and wait for a signal from the indexer or rotary table that it has finished its programmed move, then it will resume machining at the next event. If you want the ProtoTRAK VM to return the head to the Z retract position before moving to the next event, put the Aux 3 command in a Pause event. The ProtoTRAK VM will interpret the signal from the indexer or rotary table as a GO command and continue machining without you having to press the GO key.

7.5 Multiple Fixtures

You may run your program using up to six fixtures plus a base. A fixture is a location on your machine with a defined offset from your absolute 0. When you program an event to have a fixture, it will treat the offset as if it were absolute zero shift. The programmed X, Y and Z absolute dimensions are relative to the absolute reference for the specified fixture.

For example, say you had two vises on the table. On the first vise, you established the lower left jaw as the absolute 0. At the same time, you measured the distance between the absolute zero you just established and the lower left jaw of the other vise. You entered that measurement as an offset from your base vise (the first one) and the other vise, which is Fixture #2. Any events that you programmed using Fixture #2 would treat the lower left corner of that second vise like the absolute 0 for the X, Y and Z dimensions in the events.

Fixture offsets are handy for combining different programs together to run at the same time or to make multiple parts by repeating the events with different fixtures.

The fixture offsets are entered in the Set-up mode. There is a base fixture, called fixture number one. We recommend that Event #1 in your program use fixture number one. It doesn't have to; we just believe it is clearer that way.

7.5.1 The Default Fixture

In the program header screen, you entered a default fixture number (if you didn't, it assumed Fixture #1 as the default fixture). If there are program events already in current memory when you change the multiple fixture from NO to YES, they will all receive the default fixture number automatically. When you change the default fixture number in the program header screen from one fixture to another, all the events that had the previous default fixture number will be changed to the new default fixture number.

If there are no program events in current memory when you change the multiple fixture feature from NO to YES, the prompt will be added to the end of every event you then program. The default fixture number will be assumed if you press SET without specifying a different number. If you do specify a different fixture number that fixture number will become the assumed input for subsequent events when SET is pressed.

7.5.2 Fixtures and Running the Program

To run the program, first go to the DRO mode and set absolute 0 at the base fixture, Fixture #1.

In the Run mode, the SHOW ABS displays the absolute position relative to the fixture in the event being run, that is, the absolute dimension that was programmed.

7.5.3 Editing Fixtures

With the Multiple Fixtures feature turned to YES, you may edit the fixture number in the Program Mode event by event. You may also use the Search Edit feature in the Edit Mode to change fixture numbers.

See Section 11.4 for setting up fixture offsets.

7.6 Assumed Inputs

The ProtoTRAK VM will automatically program the following when you simply press SET (either INC SET or ABS SET):

Tool Offset: If the first event with an offset, CENTER. If not the first event with an offset, the same as the last event if that event was a Mill or Arc event

Feedrate: same as last event if that event was a Mill, Arc, Pocket, Frame, or Helix

Tool #: same as last event, or Tool #1 if the first event.

RPM: same as the last event.

DRILL OR BORE: Drill

PECKS FOR DRILL: 1 peck

CONRAD: 0

You may change these assumed inputs by simply inputting the desired data when the event is programmed.

7.7 Z Rapid Positioning

Between any two events the head will always move to the higher of the Z rapid of the event just completed or the Z Rapid of the next event, unless the two events are connective (see Section 5.9). Remember, when using part geometry programming, two milling events are not connective unless they lie in the same plane.

7.8 Softkeys within Events

Once a geometry (Event) such as Mill or Bolt Hole is selected, the softkeys will change. See Figure 7.8

F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	DATA FWD	DATA BACK	DATA BOTTOM	INSERT EVENT	DELETE EVENT	

FIGURE 7.8 Soft keys used while programming an event

PAGE FWD: moves forward through the programmed events.

PAGE BACK: moves backwards through the programmed events.

DATA FWD: moves forward through the event inputs. Note, use the DATA FWD key and not a SET key when you do not want to input a value.

DATA BACK: moves backwards through the event inputs.

DATA BOTTOM: puts the Highlight on the last input.

INSERT EVENT: use this to insert a new event into the program. This new event will take the place of the one that was on the right side of the screen when you pressed the INSERT EVENT key. That previous event, and all the events that follow, increase their event number by one. For example, if you started with a program of four events, if you were to press the INSERT EVENT key while Event 3 was on the right side of the screen, the previous Event 3 would become Event 4 and the previous Event 4 would become Event 5. If you insert a Subroutine event, the event numbers will increase by one as when you insert another kind of event. If you insert a copy event, the event numbers will increase by the number of events that are copied.

DELETE EVENT: this will delete the event on the right side of the screen.

7.9 Programming Events

Once you press the appropriate GO TO soft key, you will begin to define your part as a series of Events. For the ProtoTRAK VM, an Event is a geometry, or a feature of a part.

PROG	P/N 0	TOOL# 1	3 AXIS	INCH			
EVENT 0		EVENT 1					
PROGRAM NAME	0						
SCALE	1.000						
DWELL REQUEST	NO						
AUXILIARY FUNCTION	NO						
EVENT COMMENTS	NO						
MULTIPLE FIXTURES	NO						
DIMENSION DEFINITION	PART GEO						
<div style="border: 1px solid black; background-color: green; color: white; padding: 5px; display: inline-block;">Select an event.</div>							
F1	F2	F3	F4	F5	F6	F7	F8
POSN	DRILL	BOLT HOLE	MILL	ARC	POCKET	PROFILE	MORE

FIGURE 7.9.1 The header screen has been completed and is on the left side. Select an event type from the soft keys

When the MORE soft key is selected, the soft keys change to:

F1	F2	F3	F4	F5	F6	F7	F8
HELIX	SUB	COPY	THREAD MILL	PAUSE	TAP	ENGRAVE	BACK

FIGURE 7.9.2 When the More soft key is selected, these additional event types are available

After an event type is selected from the soft keys, the prompts for that event will appear on the right side of the screen. The data you need to enter to program the event will appear in the Data Input Line. As soon as you enter one piece of data by pressing the INC SET or ABS SET key, the next prompt will appear in the Data Input Line.

PROG	P/N O	TOOL# 1	3 AXIS	INCH			
EVENT 0		EVENT 1 BOLT HOLE					
PROGRAM NAME	0	DRILL OR BORE	DRILL				
SCALE	1.000	# HOLES	[REDACTED]				
DWELL REQUEST	NO	X CENTER					
AUXILIARY FUNCTION	NO	Y CENTER					
EVENT COMMENTS	NO	Z RAPID					
MULTIPLE FIXTURES	NO	Z END					
DIMENSION DEFINITION	PART GEO	RADIUS					
		ANGLE					
		# PECKS FOR DRILL					
		RPM					
		Z FEEDRATE					
		TOOL #					
# HOLES : [REDACTED]							
F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	DATA FWD	DATA BACK	DATA BOTTOM	INSERT EVENT	DELETE EVENT	

FIGURE 7.9.3 Here, a Bolt Hole event was selected. The ProtoTRAK VM is prompting you to enter the number of holes

7.10 Editing Data While Programming

While programming an event, all data is entered by pressing the appropriate numeric keys and pressing **INC SET** or **ABS SET**. If you enter an incorrect number before you press **INC SET** or **ABS SET** you may clear the number by pressing **RSTR** (Restore). Then, input the correct number and press **SET**.

If incorrect data has been entered and **SET**, you may correct it as long as you are still programming that same event. Press the **DATA BACK** or **DATA FWD** (Forward) soft key until the incorrect prompt and data are highlighted and shown in the conversation line. Enter the correct number and **SET**. The PROTOTRAK VM will not allow you to skip past prompts (by pressing **DATA FWD**) which need to be entered to complete an event except when using the A.G.E. in the Irregular Pocket or Irregular Profile event.

Previous events may be edited by pressing the **BACK** hard key to the left of the soft keys. The previous event will be shifted from the left side of the screen to the right and may be edited. The **BACK** key may be pressed all the way to the Program Header Screen (the **PAGE BACK** softkey will work as well).

7.11 LOOK

As you program each event, it is helpful to see your part drawn. For quick graphics while in the Program Mode, press the **LOOK** hard key.

This function is active at the end of each event, or whenever the conversation line is prompting Select Event. Press the **LOOK** key and the ProtoTRAK VM will draw the part. Press **LOOK** again, or **BACK** to bring back the Select Event screen. You may also select a new view or adjust the view.

Softkeys in LOOK:

ADJUST VIEW: gives additional options for adjusting the view of the drawing. See below.

FIT DRAW: automatically resizes the drawing to fit the entire part program on the screen.

LIST STEP: displays the list of events on the left side of the screen and with a purple highlight on the first event. As LIST STEP is pushed, the highlight shifts to the next event. As this happens, that event is also highlighted in the graphics by having its color change to purple.

START EVENT NUMBER: will prompt you to enter an event number for highlighting. This is useful for moving quickly to a particular event in a large program.

XY: displays a view in the XY plane.

YZ: displays a view in the YZ plane.

XZ: displays a view in the XZ plane.

3D: displays an isometric view

Softkeys in Adjust view:

FIT DRAW: automatically resizes the drawing to fit the entire part program on the screen.

▼ : shifts drawing down.

▲ : shifts drawing up.

◀ : shifts drawing to the left.

▶ : shifts drawing to the right.

ZOOM IN: makes the drawing larger.

ZOOM OUT: makes the drawing smaller

RETURN: returns you to the first LOOK screen. The adjustments you made will stay on the screen until you press another selection that overrides those adjustments. The LIST STEP function may be used with the adjustment unaltered.

Note: The LOOK routine does not check for programming errors. Use Tool Path in the SET UP Mode to check movement of the tool.

7.12 Finish Cuts

The Pocket and Profile events are designed with built-in finish cut routines because they are complete, and stand-alone pieces of geometry. Shapes machined with a series of Mill or Arc events (either with or without A.G.E. Profile) don't have an automatic routine for making finish cuts. There is, however, a very simple technique that can be used.

- a. Program the shape using the print dimensions, and ignore the need to leave material for a finish cut.
- b. Using a subroutine event, Repeat all the events in "a." but call out a different tool number.
- c. In Set-Up Mode "lie" about the tool diameter for the tool called out in events in "a.". Input a tool diameter equal to the actual tool diameter plus 2 times the finish cut you wish to leave. The PROTOTRAK VM will think the tool is bigger than it really is and, therefore, shift a little further away from the machined shape.

- d. In Set-Up Mode input the actual diameter for the tool called in the Repeat event "b". This will produce the final dimensioned cut.

7.13 Two Versus Three-Axis Programming

The ProtoTRAK VM may be operated as either a two or three-axis CNC. Many jobs in tool rooms are simply easier to do with a two-axis CNC. Many jobs are more complex or require a lot of metal removal, so the extra programming and set-up of the three-axis is worth the effort.

The ProtoTRAK VM lets you choose how much CNC you want to use on the job at hand. See Section 4.7 for switching between two and three-axis operation.

Programming is very similar between the two.

EVENT 1	BOLT HOLE	EVENT 1	BOLT HOLE
DRILL OR BORE		# HOLES	
# HOLES		X CENTER	
X CENTER		Y CENTER	
Y CENTER		RADIUS	
Z RAPID		ANGLE	
Z END		RPM	
RADIUS		TOOL #	
ANGLE			
# PECKS FOR DRILL			
RPM			
Z REEDRATE			
TOOL #			

FIGURE 7.13 Programming a Bolt Hole. On the left, the prompts required in programming in three-axis CNC. On the right, the prompts required for two-axis.

In Figure 7.13 the prompts for programming a Bolt Hole in two-axis and in three-axis are shown side by side. Note that the difference is that the three-axis requires a few additional prompts.

Rather than duplicate needlessly, this manual will define all programming in three-axis. This will serve to explain all issues in programming. For two-axis programming, some event types and prompts do not appear.

8.0 Program Mode

Part 2: Program Events

Events are fully defined pieces of geometry. By programming events, you tell the ProtoTRAK VM what geometry you want to end up with; it figures the tool path for you from your answers to the prompts and the tool information you give it in the Set-Up Mode.

8.1 POSN: Position Events

This event type positions the table and quill at a specified position. The positioning is always at rapid speed (modified by feedrate override) and in the most direct path possible from the previous location. The most common use of the position event is to move the tool around an obstacle such as a clamp. For this reason, Z and X - Y motion will not occur simultaneously. First, the Z (head) will move to the higher of the Z rapid position of the current and next event, then the X (table) and Y (saddle) will move at to the programmed position.

To program a Position event press the **POSN** soft key.

Prompts for the Position event:

X END is the X dimension to the position

Y END is the Y dimension to the position

Z Rapid is the Z dimension to the position

RPM is the spindle RPM for the position event. SET will use the RPM of the previous event.

Tool # is the tool number you assign. SET will use the tool number of the previous event.

8.2 DRILL Events

This event positions the table to the specified X and Y position, moves the HEAD at rapid to the Z RAPID location, feeds the quill to the Z END location, and rapids back to Z RAPID for drill, and feeds back for bore.

Press the **DRILL** soft key.

Prompts for the drill event:

Drill=1, Bore=2: selects whether the hole is to be drilled or bored

X: is the X dimension to the hole

Y: is the Y dimension to the hole

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the bottom of the hole

Pecks: is the number of tool withdrawal cycles (each cycle drills less and then retracts to the Z Rapid position). The factory setting is for each peck to be successively smaller, taking the largest cuts at the beginning and the smallest at the end. When the highlight is on this prompt, you may change this setting by pressing the HELP key. This will take you to a screen where you may choose to have the same amount of material taken per peck.

RPM: spindle RPM

Z Feedrate: is the drilling feedrate

Tool #: is the tool number you assign

8.3 BOLT HOLE Events

This event allows you to program a bolt hole pattern without needing to compute and program the position of each hole.

Prompts for the Bolt Hole event:

Drill=1, Bore=2: selects whether the hole is to be drilled or bored

Holes: is the number of holes in the bolt hole pattern

X Center: is the X dimension to the center of the hole pattern

Y Center: is the Y dimension to the center of the hole pattern

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the bottom of the hole

Radius: is the radius of the hole pattern from the center to the center of the holes

Angle: is the angle from the positive X axes (that is, 3 o'clock) to any hole; positive angle is measured counterclockwise from 0.000 to 359.999 degrees, negative angles measured clockwise.

Pecks: is the number of tool withdrawal cycles (each cycle drills progressively less and retracts to the Z Rapid position). The factory setting is for each peck to be successively smaller, taking the largest cuts at the beginning and the smallest at the end. When the highlight is on this prompt, you may change this setting by pressing the HELP key. This will take you to a screen where you may choose to have the same amount of material taken per peck.

RPM: spindle RPM.

Z Feedrate: is the drilling feedrate

Tool #: is the tool number you assign

8.4 MILL Events

This event allows you to mill in a straight line from any one XYZ point to another, including at a diagonal in space. It may be programmed with a CONRAD if it is connective with the next event (this next event must lie in the same plane as the Mill event).

Prompts for the Mill event:

X Begin: is the X dimension to the beginning of the mill cut

Y Begin: is the Y dimension to the beginning of the mill cut

Z Rapid: is the Z dimension to transition from rapid to feed

Z Begin: is the Z dimension to the beginning of the mill cut

X End: is the X dimension to the end of the mill cut; incremental is X Begin

Y End: is the Y dimension to the end of the mill cut; incremental is Y Begin

Z End: is the Z dimension to the end of the mill cut; incremental is Z Begin

Conrad: is the dimension of a tangential radius to the next event (that must lie in the same plane for part geometry programming)

Tool Offset: is the selection of the tool offset to right (input 1), offset to left (input 2), or tool center--no offset (input 0) relative to the programmed edge and direction of tool cutter movement and as projected in the XY plane.

RPM: spindle RPM.

Z Feedrate: is the Z feedrate from Z Rapid to Z begin

XYZ Feedrate: is the milling feedrate from Begin to End in in/min from .1 to 250, or mm/min from 5 to 6350

Tool #: is the tool number you assign

8.5 ARC Events

This event allows you to mill with circular contouring any arc (fraction of a circle) that lies in the XY plane or a vertical plane (see Section 5.3). Vertical plane arcs are also limited to those that are entirely concave or convex (in other words, if you think of the arc lying on the surface of the earth, then it can't cross the equator).

In ARC events when X Center, Y Center, and Z Center are programmed incrementally, they are referenced from X End, Y End, and Z End respectively. An ARC event may be programmed with a CONRAD if it is connective with the next event (this next event must lie in the same plane as the Arc event).

Note: When an arc is a 180° arc, there are several paths that all have the same beginning, ending, and center locations. To illustrate, Imagine that if you were on the earth's equator and you wanted to get to the other side of the earth you could go clockwise or counterclockwise around the equator, or you could go up over the north pole, or down under the south pole. The ProtoTRAK VM will automatically assume that all 180° arcs that have the same beginning, ending and center dimensions for Z, lie in the XY plane. If you want a 180° arc in a vertical plane, you must program two 90° arcs or some equivalent.

Prompts for the Arc event:

X Begin: is the X dimension to the beginning of the arc cut

Y Begin: is the Y dimension to the beginning of the arc cut

Z Rapid: is the Z dimension to transition from rapid to feed

Z Begin: is the Z dimension to the beginning of the arc cut

X End: is the X dimension to the end of the arc cut; incremental is from X Begin

Y End: is the Y dimension to the end of the arc cut; incremental is from Y Begin

Z End: is the Z dimension to the end of the arc cut; incremental is from Z Begin

X Center: is the X dimension to the center of the arc; incremental is from X End

Y Center: is the Y dimension to the center of the arc; incremental is from Y End

Z Center: is the Z dimension to the center of the arc; incremental is from Z End

Conrad: is the dimension of a tangential radius to the next event (which must lie in the same plane)

Direction: is the clockwise (input 1), or counterclockwise (input 2) direction of the arc as viewed looking down for an arc in the XY plane, looking from the front for a vertical plane, or looking from the right for a vertical YZ plane

Tool Offset: is the selection of the tool offset to right (input 1), offset to left (input 2), or tool center--no offset (input 0) relative to the programmed edge and direction of tool cutter movement and as projected in the XY plane

RPM: spindle RPM.

Z Feedrate: is the Z feedrate from Z Rapid to Z Begin

XYZ Feedrate: is the milling feedrate from Begin to End in in/min from .1 to 250, or mm/min from 5 to 6350

Tool #: is the tool number you assign

8.6 POCKET Event

This event selection gives you a choice between, circle pocket, rectangular pocket and irregular pocket within the XY plane.

Pockets include machining the circumference, as well as all the material inside the circumference of the programmed shape. If a finished cut is programmed, it will be made at the completion of the final pass. The cutter will arc in and arc out of the finish cut and position itself the finish cut dimension away from the part before moving the tool out of the part.

The factory setting for tool stepover while machining a pocket is 70%. This may be changed. When you first enter the pocket event, the blue ? will appear next to the help key. Pressing Help will give you the choice of entering a new tool stepover percentage. The value you enter here will remain the same until you change it again.

8.6.1 Circular Pocket

Press the **CIRCLE PCKT** soft key if you wish to mill a circular pocket.

Prompts for the circle pocket:

X Center: is the X dimension to the center of the circle

Y Center: is the Y dimension to the center of the circle

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the Z dimension at the bottom of the pocket; incremental is from the previous event

Radius: is the finish radius of the circle

Direction: is the clockwise (input 1), or counterclockwise (input 2) direction for milling

Passes: number of cycles to machine to the final depth spaced equally from Z Rapid to Z End (hint: keep Z Rapid small)

Entry mode: choose between a zigzag ramp and a plunge. The plunge will machine straight down Z to the appropriate Z depth. The zigzag ramp will move in a zigzag pattern to depth. See Section 8.6.5 for more information about the zigzag ramp.

Fin Cut: is the width of the finish cut. If 0 is input, there will be no finish cut. See Section 8.6.7 for a bottom finish cut.

RPM: spindle RPM.

Fin RPM: the spindle RPM for the finish cut

Z Feedrate: is the Z feedrate from Z rapid to Z end

XYZ Feedrate: is the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

Fin Feedrate: is the milling feedrate for the finish cut

Tool #: is the tool number you assign

8.6.2 Rectangular Pocket

Press **RECTANGLE** soft key if you wish to mill a rectangular pocket (all corners are 90° right angles and the sides are parallel to the X and Y axes).

The prompts for the rectangular pocket:

X1: is the X dimension to any corner

Y1: is the Y dimension to the same corner as X1

X3: is the X dimension to the corner opposite X1; incremental is from X1

Y3: is the Y dimension to the same corner as X3; incremental is from Y1

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the Z dimension at the bottom of the pocket; incremental is from the previous event

Conrad: is the value of the tangential radius in each corner

Direction: is the clockwise (input 1), or counterclockwise (input 2) direction for milling

Passes: is the number of cycles to machine to the final depth spaced equally from Z Rapid to Z End (hint: keep Z Rapid small)

Entry mode: choose between a zigzag ramp and a plunge. The plunge will machine straight down Z to the appropriate Z depth. The zigzag ramp will move in a zigzag pattern to depth. See Section 8.6.5 for more information about the zigzag ramp.

Fin Cut: is the width of the finish cut. If 0 is input there will be no finish cut. See Section 8.6.7 for a bottom finish cut.

Fin RPM: the spindle RPM for the finish cut

RPM: spindle RPM.

Z Feedrate: is the Z feedrate from Z rapid to Z end

XYZ Feedrate: is the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

Fin Feedrate: is the milling feedrate for the finish cut

Tool #: is the tool number you assign

8.6.3 Irregular Pocket

Press the **IRREG PCKT** soft key if you wish to mill a pocket other than a rectangle or circle. The Irregular Pocket event gives you the powerful Auto Geometry Engine to define a shape made up of straight lines (Mills) and arcs.

The first screen in an irregular pocket event will define the beginning point and some of its general parameters. The last event of the irregular pocket must end at the same point as defined in the first event.

X Begin: is the X dimension of the beginning of the pocket

Y Begin: is the Y dimension of the beginning of the pocket

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the Z dimension of the depth of the pocket.

Passes: is the number of cycles to machine to the final depth spaced equally from Z rapid to Z end (hint: keep Z Rapid small)

Entry mode: choose between a zigzag ramp and a plunge. The plunge will machine straight down Z to the appropriate Z depth. The zigzag ramp will move in a zigzag pattern to depth. See Section 8.6.5 for more information about the zigzag ramp.

Z Feedrate: is the Z feedrate from Z rapid to Z end

XYZ Feedrate: is the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

Fin Cut: is the width of the finish cut. If 0 is input there will be no finish cut. See Section 8.6.7 for a bottom finish cut.

Fin Feedrate: is the finish cut milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

RPM: spindle RPM.

Fin RPM: spindle RPM for the finish cut

Tool #: is the tool number you assign

When the initial screen is complete, you will define the perimeter of the pocket with a series of A.G.E. Mills and A.G.E. Arcs. Programming with the Auto Geometry Engine is explained in Section 9.0.

No islands may exist in an irregular pocket.

8.6.4 Tool Path in Pocket Events

In Program Run, the pocket path will be either the plunge or zigzag cuts to Z depth along either the X or Y, followed by the required number of cuts to clear out the interior material, and then the rough cut along the inside of the perimeter. This will be repeated for each pass and then followed by a finish pass (if FIN CUT was not zero) along the inside of the perimeter at the Finish Feedrate and final depth. If a bottom finish cut was programmed, it will be machined before the perimeter finish cut.

Whether the cuts to clear the interior material of the irregular pocket are along the X or Y-axis depends on if there are hidden areas of the pocket. The ProtoTRAK VM always looks to cut along the X-axis first. If there are areas that are hidden to the X-axis, it will machine along the Y-axis. If there are hidden areas that cannot be machined continuously in the X or Y-axis, the tool will return to Z retract and then reposition to machine the hidden area.

8.6.5 Zigzag Z Depth Cuts

In programming pocket events, you have a choice to program the cuts to Z depth either as a plunge or a zigzag ramp. For rectangular and circular pockets, the tool will start in the center of the pocket. For irregular pockets, since there is no center defined, the tool will start in the lower left corner of the pocket. The direction of the ramp will be the same as the initial direction in either X or Y, depending on how the pocket is to be cut.

The tool will zigzag back and forth along the X or Y over a length of one tool radius while at the same time moving in the Z direction. When it travels one tool radius along this direction, it will have traveled a distance of ten percent of the tool diameter along the Z. This works out to roughly ramping into the part at an angle of 11 degrees.

In order to use a zigzag ramp, the X or Y move must be larger than the diameter of the tool plus the radius of the tool, minus the finish cut of the pocket. The formula is:

$$\text{the pocket } x \text{ or } y \text{ move} > \text{tool diameter} + \text{tool radius} - \text{fin cut}$$

If the tool is too large for the zigzag ramp, the ProtoTRAK VM will give an error message during program run and will then default to plunge. This will occur for each pass of the pocket depth.

8.6.6 Conrad in Pocket Events

A conrad may be added to the last event of an Irregular Pocket. The conrad will be inserted between the end of the last event and the beginning of the next event.

8.6.7 Bottom Finish Cut

The standard finish cut is along the walls of the part, but you may have the ProtoTRAK machine a finish cut along the bottom as well. When the highlight is on the Fin Cut prompt, the blue ? appears next to the Help key. Pressing help gives you the ability to choose a Finish cut in Z. You can remove the bottom finish cut by placing the highlight on the Fin Cut prompt and pressing Help again. When you select Yes to the bottom finish cut, the following prompt will appear:

Z FIN CUT: the finish cut at the bottom.

8.7 Islands

Within the Pocket event choices, you may also select a circular, rectangular or irregular island. An island is a shape that is left standing when the surrounding material is removed. The ProtoTRAK gives you the ability to machine almost any shape as an island within a rectangular pocket. Both the shape of the island and the dimension of the surrounding pocket are defined within the island event.

The tool path for machining the island event is that the tool will first plunge or ramp into the material next to the island, offset by the programmed finish cut, to the depth of the first pass. The tool will machine the perimeter of the island, offset by the island finish cut. Then the tool will machine the material in the pocket in a spiral path, moving away from the island in the programmed clockwise or counterclockwise direction. It will continue this outward spiral motion until it encounters the programmed rectangular perimeter (or pocket). It will then follow the perimeter, offset by the pocket finish cut.

It will proceed in this manner through the number of programmed passes. On the final pass, it will machine the island finish cut, then the pocket finish cut. If a Z finish cut is programmed, it will do this in the same spiral pattern as the roughing passes between machining the island and pocket finish cuts. The tool will ramp away from the pocket finish cut by the amount of the finish cut before it rises out of the part.

8.7.1 Circular Island

Press the Circle Island soft key if you wish to mill a circular island.

Prompts for the Circle Pocket:

X CENTER: is the dimension of the center of the Island

Y CENTER: is the dimension of the center of the Island

Z RAPID: is the Z dimension of the transition from rapid to feed

Z END: is the Z dimension at the bottom of the pocket; incremental is from the previous event

RADIUS: is the finish radius of the Island

DIRECTION: is the milling direction, clockwise or counterclockwise

#PASSES: the number of roughing passes to the depth

ENTRY MODE: choose between zigzag ramp and plunge. The plunge will machine straight down Z to the appropriate Z depth. The zigzag ramp will move in a zigzag pattern to depth. See the previous section for more information about the zigzag ramp.

FIN CUT ISL: Finish cut for the Island. If 0 is input, there will be no finish cut. See the previous section for a bottom finish cut.

X1 POCKET: X dimension for one corner of the rectangular pocket that surrounds the island.

Y1 POCKET: Y dimension for one corner of the rectangular pocket that surrounds the island.

X3 POCKET: X dimension for the opposite corner of the rectangular pocket that surrounds the island.

Y3 POCKET: Y dimension for the opposite corner of the rectangular pocket that surrounds the island.

CONRAD PCKT: the value of the tangential radius in the corners of the rectangular pocket that surrounds the island.

FIN CUT PCKT: finish cut along the perimeter of the pocket. If 0 is input, there will be no finish cut. See the previous section for a bottom finish cut.

RPM: spindle RPM

FIN RPM: spindle RPM for both island and pocket finish cuts

Z FEEDRATE: is the Z feedrate from Z rapid to Z end.

XYZ FEEDRATE: the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

FIN FEEDRATE: the finish milling feedrate for both the island and pocket finish cuts

TOOL #: is the tool number you assign.

8.7.2 Rectangular Island

Press the RECT ISLAND softkey if you wish to machine a rectangular island.

Prompts for the RECT ISLAND:

X1 ISLAND: X dimension for one corner of the rectangular island.

Y1 ISLAND: Y dimension for one corner of the rectangular island.

X3 ISLAND: X dimension for the opposite corner of the island.

Y3 ISLAND: Y dimension for the opposite corner of the island.

Z RAPID: is the Z dimension of the transition from rapid to feed

Z END: is the Z dimension at the bottom of the pocket; incremental is from the previous event

CONRAD ISL: the value of the tangential radius in the corners of the island.

DIRECTION: is the milling direction, clockwise or counterclockwise

#PASSES: the number of roughing passes to the depth

ENTRY MODE: choose between zigzag ramp and plunge. The plunge will machine straight down Z to the appropriate Z depth. The zigzag ramp will move in a zigzag pattern to depth. See the previous section for more information about the zigzag ramp.

FIN CUT ISL: Finish cut for the Island. If 0 is input, there will be no finish cut. See the previous section for a bottom finish cut.

X1 POCKET: X dimension for one corner of the rectangular pocket that surrounds the island.

Y1 POCKET: Y dimension for one corner of the rectangular pocket that surrounds the island.

X3 POCKET: X dimension for the opposite corner of the rectangular pocket that surrounds the island.

Y3 POCKET: Y dimension for the opposite corner of the rectangular pocket that surrounds the island.

CONRAD PCKT: the value of the tangential radius in the corners of the rectangular pocket that surrounds the island.

FIN CUT PCKT: finish cut along the perimeter of the pocket. If 0 is input, there will be no finish cut. See the previous section for a bottom finish cut.

RPM: spindle RPM

FIN RPM: spindle RPM for both island and pocket finish cuts

Z FEEDRATE: is the Z feedrate from Z rapid to Z end.

XYZ FEEDRATE: the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

FIN FEEDRATE: the finish milling feedrate for both the island and pocket finish cuts

TOOL #: is the tool number you assign.

8.7.3 Irregular Island

Press the IRREG ISLAND key if you wish to mill an island other than a rectangle or circle. The Irregular Island gives you the powerful Auto Geometry Engine to define a shape made up of straight lines and arcs.

The first screen in an Irregular Island event will define the beginning point and some of its general parameters. The last event of the irregular pocket must end at the same point as defined in the first event.

Prompts for the Irregular Island event:

X BEGIN: X dimension to the beginning of the island.

Y BEGIN: Y dimension to the beginning of the island.

Z RAPID: is the Z dimension of the transition from rapid to feed

Z END: is the Z dimension at the bottom of the pocket; incremental is from the previous event

#PASSES: the number of roughing passes to the depth

ENTRY MODE: choose between zigzag ramp and plunge. The plunge will machine straight down Z to the appropriate Z depth. The zigzag ramp will move in a zigzag pattern to depth. See the previous section for more information about the zigzag ramp.

FIN CUT ISL: Finish cut for the Island. If 0 is input, there will be no finish cut. See the previous section for a bottom finish cut.

X1 POCKET: X dimension for one corner of the rectangular pocket that surrounds the island.

Y1 POCKET: Y dimension for one corner of the rectangular pocket that surrounds the island.

X3 POCKET: X dimension for the opposite corner of the rectangular pocket that surrounds the island.

Y3 POCKET: Y dimension for the opposite corner of the rectangular pocket that surrounds the island.

CONRAD PCKT: the value of the tangential radius in the corners of the rectangular pocket that surrounds the island.

FIN CUT PCKT: finish cut along the perimeter of the pocket. If 0 is input, there will be no finish cut. See the previous section for a bottom finish cut.

RPM: spindle RPM

FIN RPM: spindle RPM for both island and pocket finish cuts

Z FEEDRATE: is the Z feedrate from Z rapid to Z end.

XYZ FEEDRATE: the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

FIN FEEDRATE: the finish milling feedrate for both the island and pocket finish cuts

TOOL #: is the tool number you assign.

When the initial screen is complete, you will define the perimeter of the island with a series of A.G.E. Mills and A.G.E. Arcs. Programming with the Auto Geometry Engine is explained in Section 9.0.

8.8 PROFILE Events

This event allows you to mill around the outside or inside of a circular or rectangular frame or an irregular profile. The irregular profile may be closed or open. All profiles are limited to the XY plane.

When the irregular profile event is started the ProtoTRAK VM will automatically initiate the powerful Auto Geometry Engine. See Section 9.0 for programming with A.G.E.

8.8.1 Circle profile

Press the **CIRCLE** soft key if you wish to mill a circular frame.

Prompts in the Circle Profile event:

X Center: is the X dimension to the center of the circle

Y Center: is the Y dimension to the center of the circle

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the Z dimension to the bottom of the frame; incremental is from the previous event

Radius: is the finish radius of the circle

Direction: is the clockwise (input 1), or counterclockwise (input 2) direction for milling

Tool Offset: is the selection of the tool offset to the right (input 1), offset to the left (input 2), or tool center--no offset (input 0) relative to the programmed edge and direction of the cutter movement

Passes: is the number of cycles to machine to the final depth spaced equally from Z Rapid to Z End (hint: keep Z Rapid small)

Fin Cut: is the width of the finish cut. If 0 is input, there will be no finish cut

RPM: spindle RPM

Fin RPM: the spindle RPM for the finish cut

Z Feedrate: is the Z feedrate from Z rapid to Z end

XYZ Feedrate: is the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

Finish Feedrate: is the milling feedrate for the finish cut

Tool #: is the tool number you assign

8.8.2 Rectangular Profile

Press the **RECTANGLE** soft key if you wish to mill a rectangular frame (all corners are 90° right angles).

Prompts for the rectangular profile:

X1: is the X dimension to any corner

Y1: is the Y dimension to the same corner as X1

X3: is the X dimension to the corner opposite X1; incremental is from X1

Y3: is the Y dimension to the same corner as X3; incremental is from Y1

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the Z dimension at the bottom of the frame; incremental is from the previous event

Conrad: is the value of the tangential radius in each corner

Direction: is the clockwise (input 1), or counterclockwise (input 2) direction for milling

Tool Offset: is the selection of the tool offset to the right (input 1), offset to the left (input 2), or tool center--no offset (input 0) relative to the programmed edge and direction of the cutter movement

Passes: is the number of cycles to machine to the final depth spaced equally from Z Rapid to Z End (hint: keep Z Rapid small)

Fin Cut: is the width of the finish cut. If 0 is input, there will be no finish cut

RPM: spindle RPM

Fin RPM: the spindle for the finish cut

Z Feedrate: is the Z feedrate from Z rapid to Z end

XYZ Feedrate: is the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

Fin Feedrate: is the milling feedrate for the finish cut (if programmed).

Tool #: is the tool number you assign

8.8.3 Irregular Profile

Press the **IRREG PROFILE** soft key if you wish to mill a profile other than a rectangle or circle. The Irregular Profile event gives you the powerful Auto Geometry Engine to define a shape made up of straight lines (Mills) and arcs.

The Irregular Profile is a series of events that are programmed to machine continuously. The first event of the series will be called an IRR PROFILE and it will define the beginning point of the profile and other information that applies to the entire profile.

X Begin: is the X dimension of the beginning of the profile

Y Begin: is the Y dimension of the beginning of the profile

Z Rapid: is the Z dimension to transition from rapid to feed

Z End: is the Z dimension of the depth of the profile

Tool Offset: is the selection of the tool offset to right (input 1), offset to left (input 2), or tool center--no offset (input 0) relative to the programmed edge and direction of tool cutter movement

Passes: is the number of cycles to machine to the final depth spaced equally from Z rapid to Z end (hint: keep Z Rapid small)

Z Feedrate: is the Z feedrate from Z rapid to Z end

XYZ Feedrate: is the milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

Fin Cut: is the width of the finish cut. If 0 is input there will be no finish cut

Fin Feedrate: is the finish cut milling feedrate in in/min from .1 to 250, or mm/min from 5 to 6350

RPM: spindle RPM

Fin RPM: spindle RPM for the finish cut

Tool #: is the tool number you assign

When the initial Irregular Profile screen is complete, the rest of the profile is programmed using A.G.E. Mill and A.G.E. Arc events. Programming with the Auto Geometry Engine is explained in Section 7.8.

8.9 Helix Events

The Helix Event is found after you press the MORE softkey from the Select Event screen. It allows you to machine in a circular path in the XY plane while you simultaneously move the Z-axis linearly.

Press the **HELIX** soft key.

X Center: is the X dimension to the center of rotation of the helix

Y Center: is the Y dimension to the center of rotation of the helix

Z Rapid: is the Z dimension to transition from rapid to feed

Z Begin: is the Z dimension to the beginning of the helix

Z End: is the Z dimension at the end of the helix

Radius: is the radius from the center of rotation to the helix

Angle: is the angle from the positive X axis (that is, 3 o'clock) to the starting position of the helix

Rev: is the number of revolutions in the helix, for example, 0.75 would be 270 degrees, or 3.25 would be three times around plus 90 degrees

Direction: is the clockwise (input 1) or counterclockwise (input 2) direction of the helix

Tool Offset: is the selection of the tool offset to right (input 1), offset to left (input 2), or tool center--no offset (input 0) relative to the programmed edge and direction of the cutter movement

RPM: spindle RPM

Z Feedrate: is the Z feedrate from Z rapid to Z begin

XYZ Feedrate: is the feedrate from beginning to end in in/min from .1 to 250, or mm/min from 5 to 6350

Tool #: is the tool you assign

8.10 Subroutine Events

The Subroutine Events are used for manipulating previously programmed geometry within the XY plane.

The Subroutine Event is divided into three options: Repeat, Mirror, and Rotate.

Repeat and Rotate may be connective. As long as the rules of connectivity are satisfied (see Section 5.9), the ProtoTRAK VM will continue milling between preceding and subsequent events.

REPEAT allows you to repeat an event or a group of events up to 99 times with an offset in X and/or Y and/or Z. This can be useful for drilling a series of evenly spaced holes, duplicating some machined shapes, or even repeating an entire program with an offset for a second fixture.

Repeat events may be "nested." That is, you can repeat a repeat event, of a repeat event, of some programmed event(s). One new tool number may be assigned for each Repeat Event.

MIRROR is used for parts that have symmetrical patterns or mirror image patterns. In addition to specifying the events to be repeated, you must also indicate the axis or axes (X or Y or XY are allowed) that the reflection is mirrored across. In addition, you must specify the offset from absolute zero to the line of reflection. You may not mirror another mirror event, or mirror a rotate event. Consider the figure on the right:

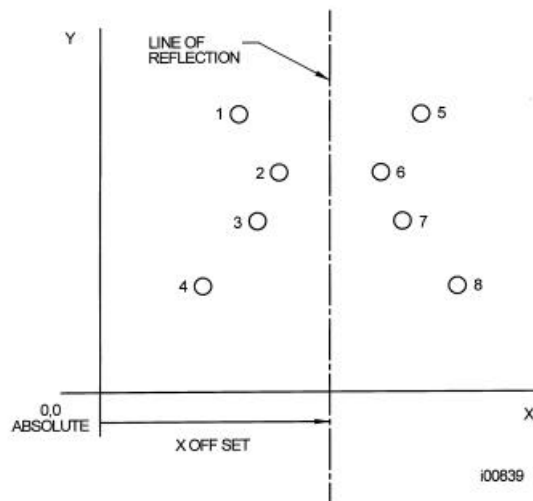


FIGURE 8.10.1 Holes 1-4 are mirrored across the Y axis to 5-8, respectively, about a line X OFFSET from X=absolute 0

ROTATE is used for polar rotation of parts that have a rotational symmetry around some point in the XY plane. In addition to specifying the events to be repeated, you must also indicate the absolute X and Y position of the center of rotation, the angle of rotation (measured counterclockwise as positive; and clockwise as negative), and the number of times the specified events are to be rotated and repeated. You may not rotate another rotate event, or rotate a mirror event. Consider the figure below:

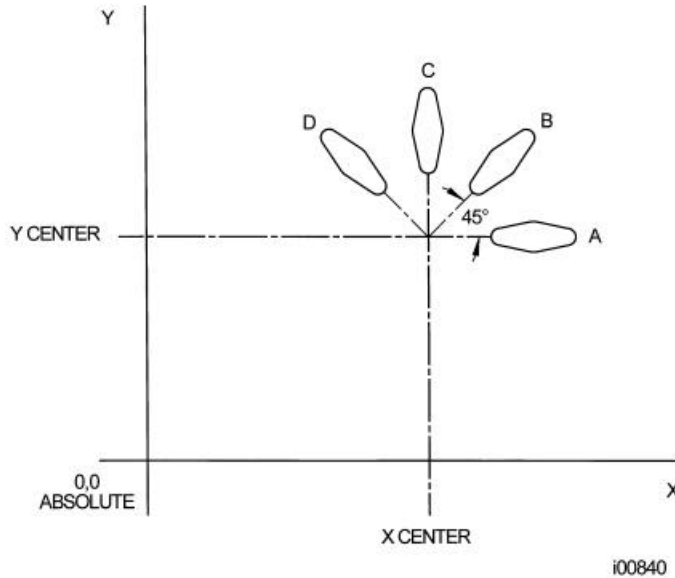


FIGURE 8.10.2 Shape A programmed with 4 MILL events and Conrads. Using **ROTATE**, these 4 events are rotated through a 45 degree angle about a point offset from absolute zero by X Center and Y Center dimensions. A is rotated 3 times to produce shape B, C, and D

Press the **SUBROUTINE** (SUB) soft key to call up the Repeat, Mirror, and Rotate options.

8.10.1 Repeat

Press the **REPEAT** soft key.

Where:

First Event #: is the event number of the first event to be repeated

Last Event #: is the event number of the last event to be repeated; if only one event is to be repeated, the Last Event # is the same as the First Event #

X Offset: is the incremental X offset from event to be repeated

Y Offset: is the incremental Y offset from event to be repeated

Z Offset: is the incremental Z offset from event to be repeated

Z Rapid Offset: is the incremental Z rapid offset from event to be repeated

Repeats: is the number of times events are to be repeated up to 99

% RPM: the percentage of the RPM programmed in the repeated events. 100% is assumed

% Feed: the percentage of the feeds programmed in the repeated events. 100% is assumed

Tool #: is the tool number you assign

8.10.2 Mirror

Press the **MIRROR** soft key.

First Event #: is the event number of the first event to be mirrored

Last Event #: is the event number of the last event to be mirrored; if only one event is to be mirrored, the last event is the same as the first.

Cutting Order: input 1 to cut from the lowest mirrored event to the highest (forward) and 2 to machine from the highest mirrored event to the lowest (backward). This way you can keep all the machine motion in a consistent direction as it moves from the original shape to the mirrored shape and keep all cutting either climb or conventional.

Mirror Axis: is the selection of the axis or axes to be mirrored (input X or Y or XY, SET)

X Offset: is the distance from Y absolute 0 to the Y-axis line of reflection

Y Offset: is the distance from X absolute 0 to the X-axis line of reflection

8.10.3 Rotate

Press the **ROTATE** soft key.

First Event #: is the event number of the first event to be rotated

Last Event #: is the event number of the last event to be rotated; if only one event is to be rotated, the last event is the same as the first

X Center: is the X absolute position of the center of rotation

Y Center: is the Y absolute position of the center of rotation

Angle: is the angle of rotation of the repeated events (positive is counterclockwise; negative is clockwise)

Repeats: is the number of times events are to be rotated up to 99

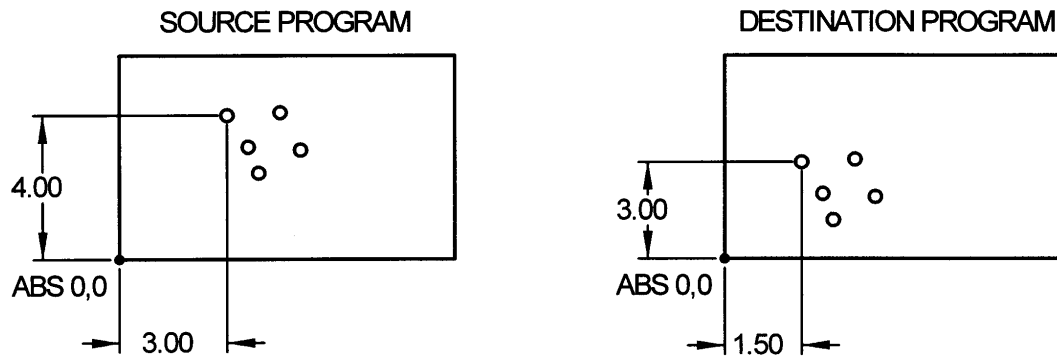
8.11 COPY Events

Copy Events are programmed exactly like Subroutine Events. The only difference is that in Copy the events are rewritten into subsequent events. If, for example, in Event 11 you Copy Repeated Events 6, 7, 8, 9, 10 with 2 repeats, Events 6-10 would be copied with the input offsets into Events 11-15, and recopied into 16-20.

Copy Events may be Repeat, Mirror, or Rotate.

Copy is very useful. With copy you can:

- Edit the events that are being repeated, mirrored or rotated without changing the original events.
- Connect repeated or rotated events to be connective so that the quill will not move up to the Z Rapid position, and back down unnecessarily. However, to be connective, you must be certain that the X, Y, Z begin of the first event, once offset or rotated, coincides with the X, Y, Z end of the last event.
- Program an event parallel to X or Y (where the geometry is the easiest to describe), rotate it to the desired position, and then delete the original.
- Use the Clipboard to paste previously stored events from another program into the current program. After you press the Clipboard key, you will enter the offset from the previous program's absolute zero to the current program's absolute zero (see figure below). For information about putting events into the clipboard, see Section 10.4.



i00965

Figure 8.11 In the above example, the offset that puts the group of holes in the desired location is $X=-1.50$ and $Y=-1.00$.

8.12 Thread Mill Event

To program a Thread Mill event press the Thread mill soft key. This event includes an automatic move in and out by 0.050" of the thread.

Prompts in the Thread Mill event:

X CENTER: the X dimension of the center of the thread

Y CENTER: the Y dimension of the center of the thread

Z RAPID: the Z dimension where the Z rapid feed slows to Z program feed

Z BEGIN: the Z dimension where the threading pass begins

Z END: the Z bottom of the thread

PITCH: the distance from one thread to the next in inches or mm. It is equal to one divided by the number of threads per inch. For example, the pitch for a 1/4-20 screw is $1 \div 20 = .05$ inches

MAJOR DIA: the largest diameter of the thread (the root for an ID thread, the crest for an OD thread)

MINOR DIA: the smallest diameter of the thread (the root for an OD thread, the crest for an ID thread)

SIDE: input 1 for inside, 2 for outside

ANGLE: the angle the tool feeds into the beginning depth

PASSES: - the number of passes to cut the thread to its final depth

RPM: spindle RPM

DIRECTION: clockwise or counterclockwise

Z FEEDRATE: The feedrate from Z Rapid to Z Begin

XYZ FEEDRATE: The feedrate of XYZ along the path of the helix.

FIN CUT - width of the finish cut. If 0 is input, there is no finish cut.

If something other than 0 is input for finish cut, the following prompts appear:

FIN RPM - the spindle RPM for the finish cut.

FIN FEEDRATE - the milling feedrate for the finish cut.

8.13 PAUSE Events

The purpose of the Pause Event is to allow you to program a stop condition within the program. The effect of this event is to turn off the spindle, move the head to the Z retract location with the X and Y position corresponding to the end of the previous event and stopping the program run.

Pause events are useful if you want to stop the program to activate an indexer (Section 7.4), make a measurement, change a fixture, etc.

*NOTE: In general, you should avoid programming a PAUSE event between two connective events (see Section 3.7). The Pause event will cause the events to **NOT** be connective.*

To program a Pause Event press the **PAUSE** soft key. Because there is no input required, simply press SET to load and the event counter will advance by one and the Select Event screen will reappear.

In run, press the **GO** key after a pause to continue.

8.14 Tap Events

Tap events allow you to tap holes using a floating tapping head. The feedrate of the thread will be calculated from the pitch and RPM entered. The maximum RPM that the ProtoTRAK VM can tap is 1000.

To program a tap event press the TAP soft key.

Prompts in the Tap event:

X: the X dimension to the center of the hole

Y: the Y dimension to the center of the hole

Z Rapid: is the Z dimension to transition from rapid to feed. Make sure that Z rapid is set high enough to compensate for the amount of float in the floating tapping head.

Z End: the depth of the thread

PITCH - the distance from one thread to the next in inches or mm. It is equal to one divided by the number of threads per inch. For example, the pitch for a 1/4-20 screw is $1 \div 20 = .05$ inches

RPM: spindle RPM

Tool #: is the tool number you assign

8.14.1 DPMV Tapping Speed Recommendations

A – aluminum only

AS – aluminum and steel

Tap Size vs RPM	200	400	600	800	1000
4-40	AS	AS	A		
8-32	AS	AS	AS	A	
¼ - 20	AS	AS	AS	AS	A
3/8 -16	AS	AS	AS	AS	A
½ -13	AS	AS	AS	A	
5/8 - 11	AS	AS	A		
¾-10	A	A			

Interpolate speeds for tap sizes in between the ones noted in the table

8.14.2 Tapping Notes and Recommendations

- Harder materials will require slower speeds and tap size may also be limited. For example, the DPMV may not be capable of tapping a 5/8-11 thread in inconel.
- Softer materials than aluminum may be capable of higher speeds and you may be able to go a little bigger in terms of tap sizes than what is listed in the table.
- ¾-10 must be run at slow speeds in soft materials only. This is at the limit in terms of tap size.
- Make sure your tap is not dull. Dull taps will require more torque to cut and may not cut threads to specification.
- Coolant will play a large role in determining the size of tap you can use in a given material. A spray mister was the lubrication used when creating the above table.
- Before tapping make sure your holder has adequate float in the tension and compression stroke. The holder should float up and down with minimal force applied to the holder. Also make sure your holder does not stick in tension (holder pulls down) or compression (holder pulls up). This will stop the tap from getting to the correct position programmed and may also break your tap.
- Always set your Z rapid higher than the tension stroke of your holder. This will save your tap if the holder does get stuck in the tension stroke between holes.
- Make sure your tap is running true in the holder.
- Most tapping problems are due to a dull tap or a holder that is not floating and is stuck in a certain position.

8.15 Engrave Event

The Engrave Event allows you to machine numbers, letters and special characters as part of a part program. See Figure 8.15 below for the letters and special characters that are available in the Engrave Event.

When programming with the Engrave Event, the ProtoTRAK will construct a box to contain the text you define. This box is oriented along the X axis like the text in this sentence, and you may program up to 40 characters per event (although you will only be able to see 20 characters on the prompts screen). To machine text in a direction other than the X axis, simply use multiple Engrave Events and place the lower left corner of the box wherever you would like. The numbers and letters you program will always have a standard orientation (like the letters on this page) – you cannot program tilted or inverted letters with the Engrave Event. The letters are of the font shown in the figure and all capitals.

8.15.1 Prompts for the Engrave Event

First, define the lower left corner of the box that will contain your text:

X BEGIN: The X coordinate of where you want your text to begin

Y BEGIN: The Y coordinate of where you want your text to begin

Z RAPID: The Z dimension where the Z rapid feed slows to Z program feed

Z END: The Z dimension to the bottom of your text.

HEIGHT: The height of your text. Each character varies in width; the set height of the character will change the width in order to keep the overall size of the character proportional.

TEXT: The text to be milled. When you get to this prompt, the Alpha keys will automatically pop up to allow you to enter the text. Once you have finished entering text, you must press End (F8) and then any of the SET keys to successfully enter your text into the event. The alpha keys will appear automatically if the text field is blank. If you have already entered text but wish to make a change, you will see a blue question mark appear on the lower left corner of the screen when you scroll to this field, press the Help button and the alpha keys will appear.

Z FEEDRATE: Is the feedrate from Z rapid to Z end

XYZ FEEDRATE: The feedrate of XYZ along the path of the text

RPM: Spindle RPM

Tool #: is the tool number you assign

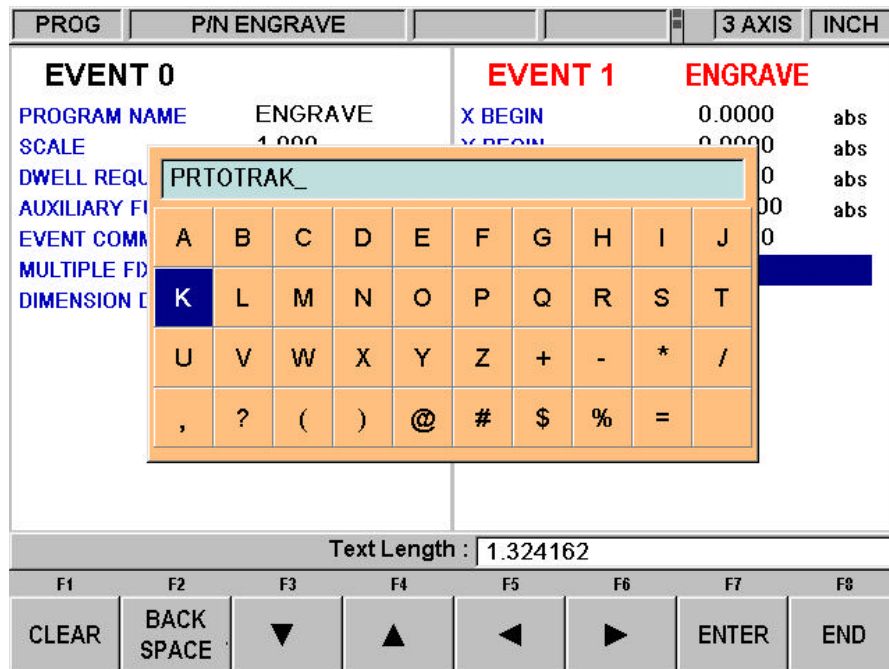


FIGURE 8.15 The above figure shows the text and special characters available for the Engrave event. Notice the field that is labeled “Text Length”. This field will display the total length of your programmed text and will update as you enter each character.

8.16 Finishing Teach Events

Teach events are either POSN, DRILL or MILL events that are originated in the DRO Mode (see Section 6.6).

The Teach events that are started in the DRO Mode must be finished in the Program Mode before running. Teach events are of these different types:

TEACH POSN - for two-axis operation, the Position and Drill event types are combined. See Section 8.1 for a description of Position event prompts.

TEACH DRILL - this may also be made into a bore event. See Section 8.2 for a description of Drill event prompts.

TEACH MILL - a straight line that specifies the beginning and the end. When TEACH MILL events are defined using the CONT MILL softkey, the prompts for information that cannot change will be suppressed. See Section 8.4 for a description of Mill event prompts.

When a Teach event is unfinished, the words NOT OK will appear next to the event type. Once the prompts are completed, the words NOT OK and Teach will disappear. The event will become a normal MILL, DRILL, or POSN event.

9.0 Program Mode

Part 3: The Auto Geometry Engine (A.G.E.) Programming

When you program an Irregular Pocket (Section 8.6.3) or an Irregular Profile (Section 8.7.3) the Auto Geometry Engine, or A.G.E. is automatically started.

The A.G.E. is powerful software that works behind the easy-to-use geometry programming of the ProtoTRAK VM. It is treated in its own section because it works differently than the other event types. Unlike other events, the A.G.E. allows you to:

- Enter the data you know, and skip the prompts you don't.
- Use different types of data (like angles) that may be available from the print.
- Enter guesses for the X and Y ends and centers not available on the print.

With the A.G.E., you can easily overcome limitations in the data the print provides without having to spend time in laborious calculations.

9.1 Starting the A.G.E.

The A.G.E. is started automatically when you enter the Irregular Pocket or Irregular Profile event. The first set of prompts you encounter will be the header information. Once that information is entered, you will see the following screen:

PROG	P/N 0806 BRACKET			3 AXIS	INCH	
EVENT 1		IRR PROFILE		EVENT 2		
X BEGIN	0.0000	abs				
Y BEGIN	0.0000	abs				
Z RAPID	0.0500	abs				
Z END	-0.2500	abs				
TOOL OFFSET	RIGHT					
# PASSES	2					
FIN CUT	0.0100					
RPM	2000.00					
FIN RPM	2500.00					
Z FEEDRATE	4.0					
XYZ FEEDRATE	15.0					
FIN FEEDRATE	10.0					
TOOL #	1		Select an event.			
F1 F2 F3 F4 F5 F6 F7 F8						
A.G.E. MILL	A.G.E. ARC				END A.G.E.	ABORT A.G.E.

FIGURE 9.1 Once the profile header screen is finished, you choose between an A.G.E. Mill and an A.G.E. arc to define the shape

Where:

A.G.E. Mill: A straight line from one X Y point to another.

A.G.E. Arc: Any part of a circle.

End A.G.E.: Ends the A.G.E. programming for the Irregular Pocket or Irregular Profile.

Abort A.G.E.: Aborts all A.G.E. events. The data for all the events is lost.

9.2 A.G.E. Mill Prompts

Press the A.G.E. Mill key.

PROG	P/N 0807-5		3 AXIS	INCH			
EVENT 1	IRR PROFILE		EVENT 2	A.G.E. MILL Not OK			
X BEGIN	-2.0000 abs		TANGENT	NO			
Y BEGIN	0.0000 abs		X END	0.0000 abs			
Z RAPID	0.0200 abs		Y END				
Z END	-0.3500 abs		CONRAD				
TOOL OFFSET	RIGHT		ANGLE END				
# PASSES	2		LENGTH				
FIN CUT	0.0100		LINE ANGLE				
RPM	2250.00						
FIN RPM	3000.00						
Z FEEDRATE	5.0						
XYZ FEEDRATE	15.0						
FIN FEEDRATE	10.0						
TOOL #	1						
X END : 0.0000 abs							
F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	DATA FWD	DATA BACK	DATA BOTTOM	INSERT EVENT	DELETE EVENT	GUESS

FIGURE 9.2 A.G.E. Mill prompts. Enter what you know, skip or guess the ones you don't

Prompts in A.G.E. Mill programming:

Tangent: this refers to the tangency of the mill to the previous event. See Section 9.11 for a discussion of tangency.

X END: is the X dimension to the end of the mill cut; incremental is X Begin

Y END: is the Y dimension to the end of the mill cut; incremental is Y Begin

CONRAD: is the dimension of a tangential radius to the next event

ANGLE END: is the angle measured counterclockwise from this mill event to the next. Do not input if the next event is an arc

LENGTH: is the length of the mill from beginning to end

LINE ANGLE: is the angle of this mill line (moving from begin to end) measured counterclockwise from the positive X axis (that is 3 o'clock)

GUESS: This softkey will appear when the prompt is on X or Y dimensioned data. Press the Guess key before you press INC SET or ABS SET to enter the data as a guess. See Section 9.7

9.3 A.G.E. Arc Prompts

Press the A.G.E. ARC key.

Prompts in A.G.E. Arc programming:

Tangent: this refers to the tangency of the mill to the previous event. See Section 9.11 for a discussion of tangency.

DIRECTION: is the clockwise (input 1), or counterclockwise (input 2) direction of the arc

X END: is the X dimension to the end of the arc cut; incremental is from X Begin

Y END: is the Y dimension to the end of the arc cut; incremental is from Y Begin

X CENTER: is the X dimension to the center of the arc; incremental is from X End

Y CENTER: is the Y dimension to the center of the arc; incremental is from Y End

CONRAD: is the dimension of a tangential radius to the next event

RADIUS: is the radius of the arc

CHORD LENGTH: is the straight line distance from the begin point to the end point

CHORD ANGLE: is the angle spanned by the arc

In addition to the normal Softkeys, this additional one will appear in A.G.E. Arc programming:

GUESS: this softkey will appear when the prompt is on X or Y dimensioned data. Press the Guess key before you press INC SET or ABS SET to enter the data as a guess. See Section 9.7

9.4 Skipping Over Prompts

In the A.G.E., events don't have to be fully defined before you can go to the next one. You can skip the data you don't know by using the DATA FWD softkey. After you press the DATA FWD key at the last prompt, the event will move to the left side of the screen and the Select Event screen will appear.

When skipping over prompts or editing, always use the DATA FWD or DATA BACK key. Using INC SET or ABS SET will change the data.

If you want the event back on the right side, use the BACK hard key.

9.5 The OK/NOT OK Flag

Each A.G.E. event has a flag that tells you if it has been fully defined. Sometimes data from later events is needed to define previous events. To the immediate right of the event type, the words OK or NOT OK appear, depending on whether that particular event is defined.

Once the OK flag appears for the event, you do not need to enter more information. Skip past the rest of the prompts with the DATA FWD softkey.

If you leave the Program Mode and then return, pressing the GO TO END softkey will take you automatically to the first NOT OK event.

9.6 Ending A.G.E.

Any time all the events of an Irregular Profile are OK, the A.G.E. may be ended. If you are programming an Irregular Pocket, there is an additional requirement that must be satisfied before the A.G.E. may be ended: the X and Y end point of the last event must be the same as the X and Y beginning point, so that the pocket is closed. Otherwise, the ProtoTRAK VM CNC cannot program the tool path to clear the pocket.

The Irregular Profile has no such restriction since profiles may be open or closed.

Once the A.G.E. is ended, the Irregular Pocket or Irregular Profile event is complete and you may then choose from all the programming canned cycles from the Select an Event screen. To reopen the A.G.E. Profile or Pocket, simply use the BACK hard key or the PAGE FWD or PAGE BACK softkeys to position on of the A.G.E. events on the right side of the screen. You may edit or insert other events.

9.7 Guessing Data

Whenever you are missing X or Y Ends or Centers, you should generally enter a guess. Gussed data is treated differently by the ProtoTRAK VM than regular data. Often, the information you put into the system will allow it to calculate a mathematically correct line or arc that would satisfy the conditions of the hard data you entered. This line or arc may yield more than one solution to particular point you are looking for. That is where the Guess comes in: the A.G.E. uses the guess to choose from the mathematically possible solutions. In most cases, your guesses do not have to be very precise. The smaller the lines or arcs, the more precise the guess should be.

PROG	P/N 0807-5			3 AXIS	INCH		
EVENT 1	IRR PROFILE		EVENT 2	A.G.E. MILL	Not OK		
X BEGIN	-2.0000 abs		TANGENT	YES			
Y BEGIN	0.0000 abs		X END	G 1.0000 abs			
Z RAPID	0.0200 abs		Y END				
Z END	-0.3500 abs		CONRAD				
TOOL OFFSET	RIGHT		ANGLE END				
# PASSES	2		LENGTH				
FIN CUT	0.0100		LINE ANGLE				
RPM	2250.00						
FIN RPM	3000.00						
Z FEEDRATE	5.0						
XYZ FEEDRATE	15.0						
FIN FEEDRATE	10.0						
TOOL #	1						
Guess for Y END : <input type="text"/>							
F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	DATA FWD	DATA BACK	DATA BOTTOM	INSERT EVENT	DELETE EVENT	GUESS

FIGURE 9.7 The X End dimension has been entered as a guess—note the letter G

Guesses should always be entered as absolute dimensions. Once entered, the guessed data is green and there is a 'G' next to it. Gussed data will be labeled this way in all the events that are flagged NOT OK. Once an event is OK, the guessed data will be replaced

by calculated data. If you wish to edit your guesses, placing it on the right side of the screen will cause your original guessed data to reappear.

9.8 LOOK and Guess

Guessed data may be entered by pressing the number keys and then SET. However, you may find it more convenient to use the LOOK graphics to enter guesses.

When the highlight is on the prompt for which you wish to enter a guess, press the Guess key. The Data Input Line will say "Enter Guess for X END" (for example). At this point, press the LOOK key.

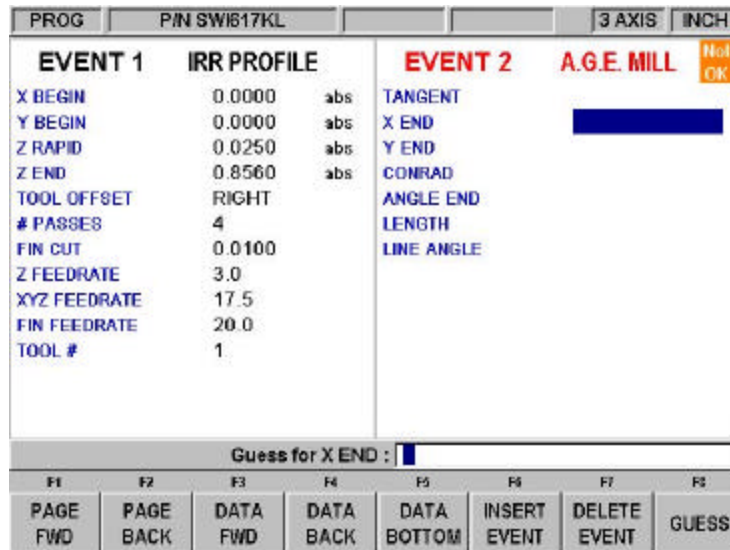


Figure 9.8.1 When the Data Input Line says "Enter Guess" pressing LOOK gives you the ability to use graphics to make your guesses.

On the screen shown in the figure above, the Data Input Line says "Enter Guess for X BEG". Pressing LOOK at this point will take you to a special version of the LOOK graphics. Using a mouse or the cursor keys, you may move a point around the screen. When you come to the place where your point is, use the Enter key.

The softkeys for this special version of the LOOK graphics:

← → ↑ ↓: move the cursor around the screen.

ZOOM IN: makes the drawing larger.

ZOOM OUT: makes the drawing smaller.

ENTER END: when the cursor is at the point you want to use as a guess, use this to enter the end point of a line or an arc.

ENTER CENTER: use this to enter a guess for the center of an arc.

You can enter a combination of guessed and non-guessed data. For example, if you were to enter the dimension for X End without guessing, you would still be able to enter the dimension of Y End using guess.

Your guess entries are loaded into the program when you exit the LOOK screen by pressing BACK or by pressing LOOK again. The ProtoTRAK will use the last ENTER key press and load that into the program.

When you use the graphics to guess dimensions on arcs, you may load in guesses for both the X/Y End and the X/Y Center before leaving the LOOK screen.

When you have not first pressed the Guess key, pressing LOOK gives you the same screen as in regular programming. Whether you enter the guesses as key presses or by using the graphics, the drawing of the LOOK screen distinguishes between events that are fully defined and those that rely on guessed data. OK events are represented by solid lines. NOT OK events are represented by dashed lines.

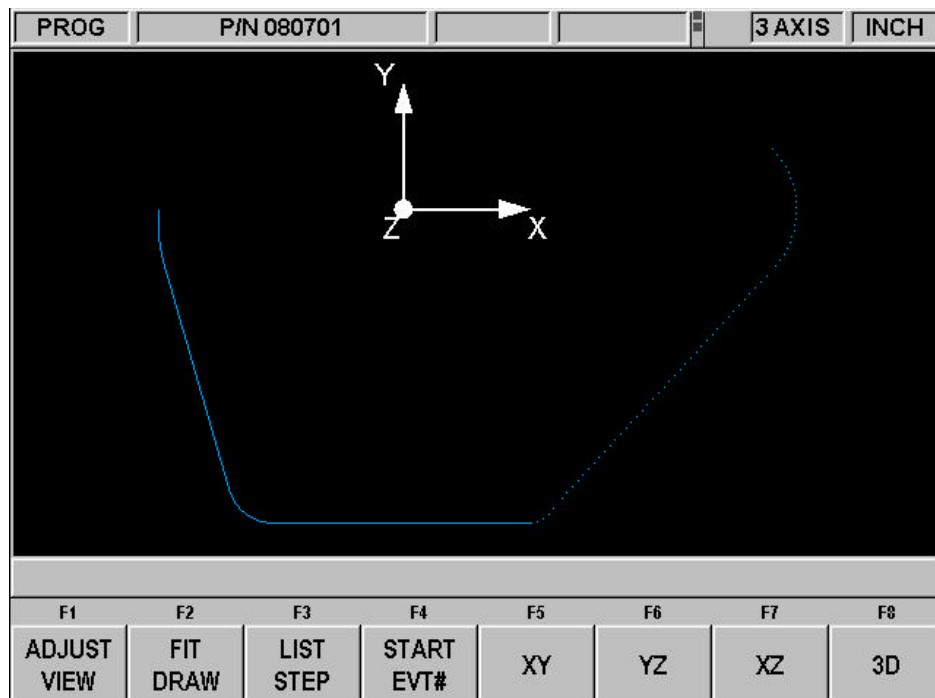


FIGURE 9.8.2 when the events are calculated based on Gussed data, they are represented by a dotted line

9.9 Calculated Data

Prompts that are skipped or for which guesses are entered may be replaced by data calculated by the ProtoTRAK VM. Calculated data is shown in red in order to distinguish it from the data that you entered. You cannot edit calculated data, but you may edit your original input. By putting the event with the calculated data on the right side of the screen, you may position the cursor to the prompt and re-input the data.

9.10 Arcs and Conrads

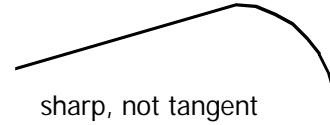
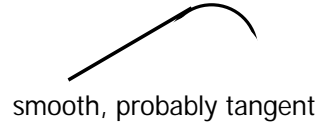
If the print is missing a lot of data, it may be desirable to program arcs as separate events where possible. This gives the system more information to work with.

9.11 Tangency

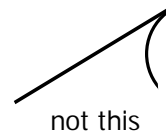
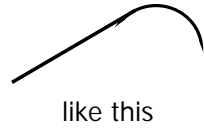
Tangency can occur between a mill and arc or an arc and arc. Specifically it means that the two events share one and only one point. You would answer yes to the TANGENCY

prompt if the event you are programming is tangent to the **previous** event. The information that events are tangent helps the Auto Geometry Engine calculate other dimensions.

You can often tell by looking at the print if events are tangent: tangent intersections tend to blend smoothly, without a sharp corner.



For the A.G.E., the tangent mill or arc are assumed to continue in the same direction, and not double back on the previous event:



10.0 Edit Mode

Within Program Mode, you can recall and re-input specific data prompt by prompt. The Edit Mode contains powerful routines for more extensive program changes.

The changes you make in the Edit Mode affect only the program in current memory. In order to preserve the changes for future use, the program must be stored again under the same name in the In/Out Mode.

10.1 Delete Events

To delete a group of events in the program, press Delete Events.

The Data Input Line will prompt for the first event to be deleted. Input the event number of the first event and press set. Next the Data Input Line will prompt for the last event number to be deleted. Put in the last number and press Set.

The remaining events will be renumbered.

10.2 Spreadsheet Editing

Spreadsheet Editing allows you to view program inputs in a table and make global changes to the program. This is particularly useful if you are working with a large program and you need to make a change to many events.

When you press the SEARCH EDIT softkey, the screen will load a table that contains data for every event. See Figure 10.2.1

EDIT		P/N R0424-11				INCH	
EVT#	TYPE	TOOL#	RPM	FIN RPM	Z FEED	XYZ FEED	FIN FEED
1	POSITION	1 (0.2500)	1000.00				
2	DRILL	1 (0.2500)	1000.00		5.0		
3	BOLT HOLE	1 (0.2500)	1000.00		5.0		
4	MILL	2 (0.3750)	1000.00		5.0	10.0	
5	MILL	2 (0.3750)	1000.00		5.0	10.0	
6	MILL	2 (0.3750)	1000.00		5.0	10.0	
7	ARC	2 (0.3750)	1000.00		5.0	10.0	
8	CIRC PCKT	2 (0.3750)	1000.00		5.0	10.0	10.0
9	TAP	3 (0.1875)	500.00				
10	HELIX	3 (0.1875)	500.00		5.0	10.0	
11	THREAD MILL	4 (0.2400)	500.00		5.0	10.0	
12	POSITION	3 (0.1875)	1000.00				

? TOOL # : 1

F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	▼	▲	◀	▶	SORT	CHANGE ALL

FIGURE 10.2.1 The Search Edit softkey launches Spreadsheet Editing. View the entire program by the variables you select

The first time the screen appears, the data is sorted by event number. Each row represents the data for the event number shown in the first column on the left. The event number is always displayed in the first column, but the other data displayed on the table can be changed.

Soft Keys in Search Edit:

PAGE FWD: pages forward through the table.

PAGE BACK: pages backwards through the table.

▼ ▲ ◀ ▶ : highlights data for editing. Only data that is highlighted and appears in the Data Input Line may be edited. Note: the EVT# (event number) and (event) TYPE may not be edited in Search Edit so the highlighter will not go there.

SORT: enables you to change the sort to any of the data displayed. See Section 10.2.2

CHANGE ALL: enables you to make global changes of data. See 10.2.3

10.2.1 Selecting Data to be Displayed on the Search Edit Table

In order to change the data selected in the table, press the HELP hard key. There will be a listing of all the data types that may be edited in Search Edit. Press the RETURN soft key and the table will be reloaded with the data that you selected.

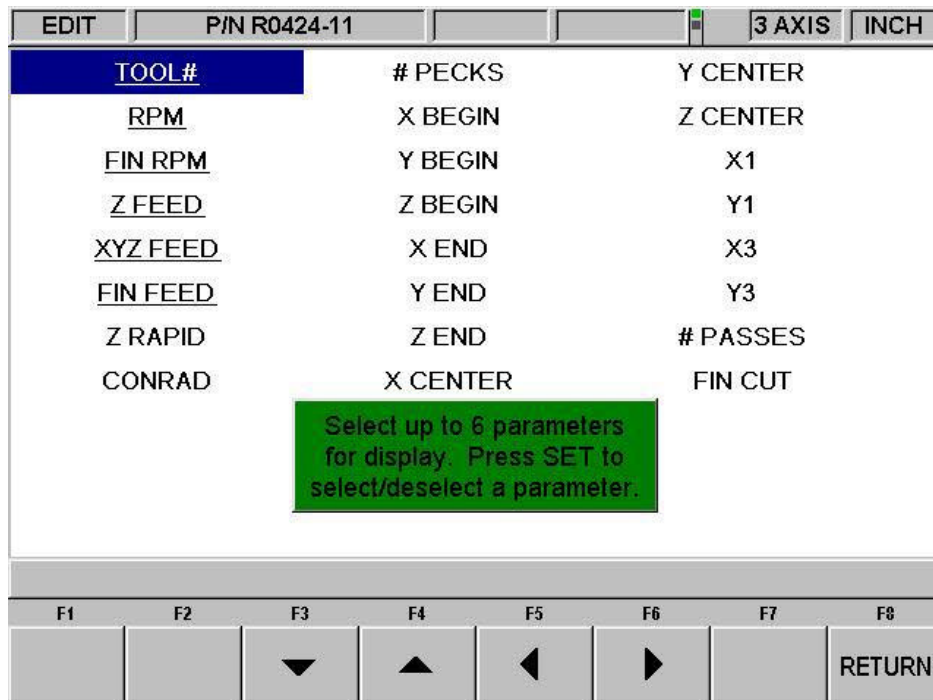


FIGURE 10.2.2 Pressing Help while viewing the spreadsheet lets you change the program parameters

After you press the HELP hard key, the screen will display all the different parameters that can be displayed on the spreadsheet. To either select or deselect any parameter, simply highlight that parameter and press SET. When you are finished, press the Return softkey and return to the spreadsheet.

10.2.2 Sorting Data

Data may be sorted by any of the data types displayed in the column head. Red letters show which column is used for sorting the data.

To change the sort, press the SORT softkey, then select the type of data you want to use for sorting from the softkeys.

The table will be changed to sort the data in ascending order (the smallest value first, the largest last).

10.2.3 Making Global Changes to Data

Sometimes it is useful to be able to change data in a program without having to go through each event one at a time. For example, if you were to want to change the tool number for every milling event, it may be a chore to go through each event in a long program to make the changes on that event type.

In order to make global changes:

1. Sort the data in a way that groups together the things you want to change.
2. Highlight the data value that is highest on the table (nearest to the top) that you want changed.
3. Press the CHANGE ALL softkey. All the inputs that are the same as the one you highlighted and are listed together below the data you highlighted will then be highlighted.
4. Enter the new value, then press set. All the highlighted data will be changed to the value you just input.

Example:

From the screen shown in Figure 10.2.1, we will change the RPM for each of the circle pockets in the program.

1. Sort by event type to get all the circle pockets together.
2. Highlight the RPM in the first circle pocket event (Event # 8). See Figure 10.2.3
3. Press the CHANGE ALL softkey. All the RPM's in the circle pocket events are highlighted. See Figure 10.2.4
4. Type in the new RPM value and press INC SET or ABS SET. See Figure 10.2.5

In this example, the RPM was changed from 1000 to 2250.

EDIT		P/N R0424-11						INCH
EVT#	TYPE	TOOL#	RPM	FIN RPM	Z FEED	XYZ FEED	FIN FEED	
38	MILL	3 (0.1875)	1000.00		5.0	10.0		
39	MILL	3 (0.1875)	1000.00		5.0	10.0		
7	ARC	2 (0.3750)	1000.00		5.0	10.0		
18	ARC	3 (0.1875)	1000.00		5.0	10.0		
29	ARC	2 (0.3750)	1000.00		5.0	10.0		
40	ARC	3 (0.1875)	1000.00		5.0	10.0		
8	CIRC PCKT	2 (0.3750)	1000.00		5.0	10.0	10.0	
19	CIRC PCKT	3 (0.1875)	1000.00		5.0	10.0	10.0	
30	CIRC PCKT	2 (0.3750)	1000.00		5.0	10.0	10.0	
41	CIRC PCKT	3 (0.1875)	1000.00		5.0	10.0	10.0	
9	TAP	3 (0.1875)	500.00					
20	TAP	3 (0.1875)	750.00					

? RPM : 1000.00

F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	▼	▲	◀	▶	SORT	CHANGE ALL

FIGURE 10.2.3 After sorting by Event Type, the highlighter is placed on the RPM of the first Circle Pocket Event

EDIT		P/N R0424-11						INCH
EVT#	TYPE	TOOL#	RPM	FIN RPM	Z FEED	XYZ FEED	FIN FEED	
38	MILL	3 (0.1875)	1000.00		5.0	10.0		
39	MILL	3 (0.1875)	1000.00		5.0	10.0		
7	ARC	2 (0.3750)	1000.00		5.0	10.0		
18	ARC	3 (0.1875)	1000.00		5.0	10.0		
29	ARC	2 (0.3750)	1000.00		5.0	10.0		
40	ARC	3 (0.1875)	1000.00		5.0	10.0		
8	CIRC PCKT	2 (0.3750)	1000.00		5.0	10.0	10.0	
19	CIRC PCKT	3 (0.1875)	1000.00		5.0	10.0	10.0	
30	CIRC PCKT	2 (0.3750)	1000.00		5.0	10.0	10.0	
41	CIRC PCKT	3 (0.1875)	1000.00		5.0	10.0	10.0	
9	TAP	3 (0.1875)	500.00					
20	TAP	3 (0.1875)	750.00					

RPM : 1000.00

F1	F2	F3	F4	F5	F6	F7	F8
							RETURN

FIGURE 10.2.4 Pressing the Change All softkey highlights the RPM for all the Circle Pocket events

EDIT		P/N R0424-11			INCH		
EVT#	TYPE	TOOL#	RPM	FIN RPM	Z FEED	XYZ FEED	FIN FEED
38	MILL	3 (0.1875)	1000.00		5.0	10.0	
39	MILL	3 (0.1875)	1000.00		5.0	10.0	
7	ARC	2 (0.3750)	1000.00		5.0	10.0	
18	ARC	3 (0.1875)	1000.00		5.0	10.0	
29	ARC	2 (0.3750)	1000.00		5.0	10.0	
40	ARC	3 (0.1875)	1000.00		5.0	10.0	
8	CIRC PCKT	2 (0.3750)	2250.00		5.0	10.0	10.0
19	CIRC PCKT	3 (0.1875)	2250.00		5.0	10.0	10.0
30	CIRC PCKT	2 (0.3750)	2250.00		5.0	10.0	10.0
41	CIRC PCKT	3 (0.1875)	2250.00		5.0	10.0	10.0
9	TAP	3 (0.1875)	500.00				
20	TAP	3 (0.1875)	750.00				
?			RPM : 2250.00				
F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	▼	▲	◀	▶	SORT	CHANGE ALL

FIGURE 10.2.5 Type the new RPM and then SET to change all the highlighted values

10.3 Erase Program

Use the ERASE PROG soft key to erase the program from the current memory. Erasing the program from current memory will not affect any programs that are stored.

If you have made changes to the program and wish to save this modified program, you will need to store it. See Section 13.4

10.4 Clipboard

The Clipboard feature is a way to copy events in one program in order to put them into a different program. It is a two-part process that takes place in two different Modes. First, in the Edit Mode, the desired events are copied, or placed on the Clipboard, from the source program. Then the events are inserted into the destination program in the Program Mode.

When you press the Clipboard key from the Edit Mode, you start the process that copies the events that you want to put into a different program than the one in current memory.

Before you do that, you should write a program or open the program file that has the events you want to copy. This is called the source program.

Inspect the events you want to copy. Make sure that the dimensioned data uses Absolute references in the first event to be copied and in all events where it will be important. Incremental references may be used, but keep in mind where the Incremental reference will be made from. See the section on Incremental Reference Position in this manual.

In addition, you may want to modify this program in order to get all the events you want together. For example, if you want to copy events 2-5 and 7-12, you may want to modify the program to delete events 1 and 6 first. That way, you can copy all of the events as they are now numbered from 1 to 10. Remember that you can modify this program just for this purpose and it will not affect the original program unless you save it with the modifications in the Program In/Out Mode.

When the source program is ready, press the CLIPBOARD softkey. A message will appear that says "Copy Events Onto Clipboard" and the Data Input Line will read "From Event". Enter the number of the first event that you want copied and press SET.

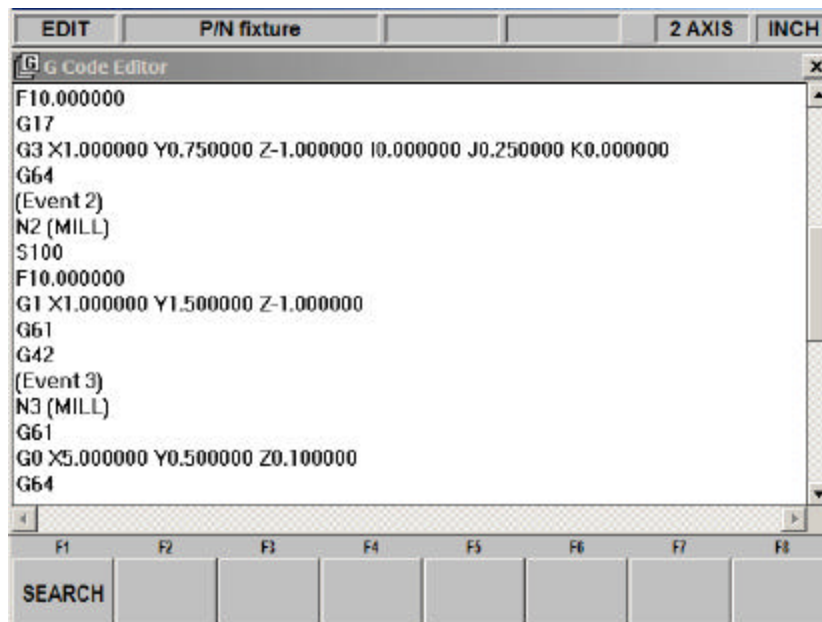
The Data Input Line will read "To Event". Enter the number of the last event you want copied and press SET.

The group of events that you have specified is now on the clipboard and will remain there until you replace it with something else by going through the same procedure. When power is turned off to the CNC the clipboard information will also be lost.

The events on the clipboard are inserted into a program in the Program Mode. See Section 8.10.

10.5 G-Code Editor

The G-Code Editor allows the edit of G-Code programs that are opened as .GCD files. Once edited, the program may be re-saved as .GCD files. ProtoTRAK Geometry-style programs may not be saved as .GCD files.



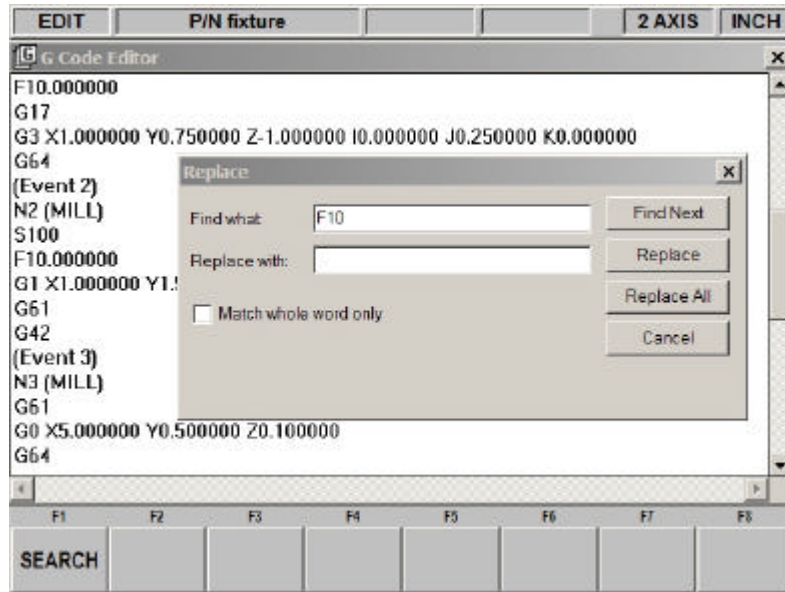
i01142

Figure 10.5.1 Use the G-Code Editor to modify G-Code programs.

You must connect a mouse and keyboard in order to use the G-Code Editor.

When you enter the G-Code Editor, the G-Code program is displayed starting at the first Block Number. Use the scroll bar to move up and down through the program. Use the mouse and keyboard to edit like you would an MS Notepad™ file.

Search allows you to launch a simple find-and-replace routine to aid in editing large G-Code files.



i0114

Figure 10.5.2 The Find and Replace routine.

Click in the Find What box and enter the item you want to find. Click on the Find Next box and the G-Code Editor will locate the next occurrence of that item. Successive clicks on Find Next will continue to search through the program. Use Match Whole Word to limit the search to the entire word. For example, if you want to find G2, but not G20 or G22, select Match Whole Word Only.

Instead of typing the item into the Find What box, you may simply highlight an item on the G-Code Editor screen. That item will be entered into the Find What box for you.

To make changes to Find What items, type what you want to have into the Replace With box. You can replace items one at a time by clicking first the Find Next box then the Replace With box for as many changes as you want to make. You can replace every item in the program with a single click of the Replace All box.

Return closes the G-Code Editor and returns the screen to the Edit Mode.

Note: If you use the USB Thumb Drive to store a G-code (.gcd) program file, you must leave the Thumb Drive plugged into the USB port the entire time the program is in current memory. If you unplug the thumb drive with the program still in current memory, the ProtoTRAK will display an error message.

11.0 Set Up Mode

The Set Up Mode contains the tool library, the tool path graphics and the machine's reference positions. Enter the Set-Up Mode by pressing the SET-UP soft key at the Select Mode screen.

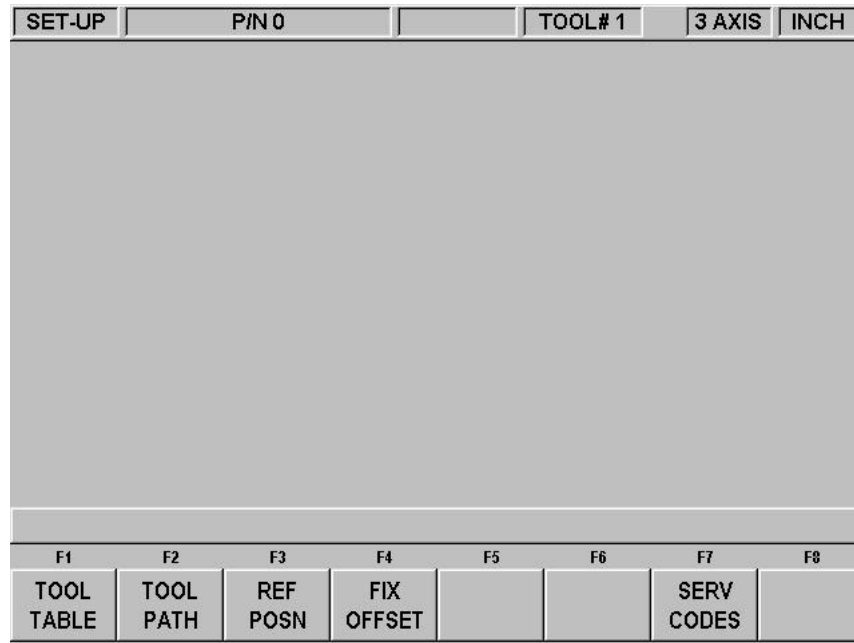


FIGURE 11.0 The Set-Up mode

11.1 The Tool Table

From the screen above, press the TOOL TABLE softkey.

SET-UP				3 AXIS	INCH		
<u>TOOL #</u>	<u>DIAMETER</u>	<u>Z OFFSET</u>	<u>Z MODIFIER</u>	<u>TOOL TYPE</u>			
REF	SET						
1	0.5000	-0.0100	0.0000	Rough End Mill			
2	0.3750	-0.8255	0.0000	Finish End Mill			
3	0.2500	-1.3320	0.0000	Drill			
4	0.3750	-0.0200	0.0000	Drill			
5	0.0000	-0.5568	0.0000	Tap			
6	0.0000	0.0000	0.0000				
7	0.0000	0.0000	0.0000				
8	0.0000	0.0000	0.0000				
9	0.0000	0.0000	0.0000				
10	0.0000	0.0000	0.0000				
11	0.0000	0.0000	0.0000				
12	0.0000	0.0000	0.0000				
13	0.0000	0.0000	0.0000				
14	0.0000	0.0000	0.0000				
15	0.0000	0.0000	0.0000				
TOOL #1 DIAMETER :				0.5000			
F1	F2	F3	F4	F5	F6	F7	F8
DATA DOWN	DATA UP	DATA LEFT	DATA RIGHT		ERASE TABLE	JOG	RETURN

FIGURE 11.1 The Tool Table

i011

11.1.1 The Tool Table Screen

When you first enter the tool table by pressing the TOOL TABLE soft key, you will see the screen shown in Figure 11.1.

Tool #: the number of the tool from 1 to 99. Tool numbers shown in red are active for the program in current memory.

Diameter: the diameter of the tool.

Z Offset: the difference between the Z position of the tool and the Z position of the reference. The Z offset is always relative to a reference point. Before the reference point is set, the highlight will not go into the Z Offset column because setting a Z offset before the Z reference is set has no meaning.

Z modifier: a value you enter to make adjustments for the tool depth. See 11.1.7 below.

Tool Type: allows you to select the type of tool from a list. input the number that corresponds to the desired name (eg, 1= drill) and press SET. The tool name will be in the prompt at the beginning of the program Run.

Ref: the reference position for the Z offset. Before the reference position is set (and the Ref row reads "NOT SET") the highlight will not go into the Z Offset column. Once it set, the highlight will not go into the Ref row, that is, you will not be able to highlight and reset your reference once it says "SET".

The soft keys in the tool table:

DATA DOWN, DATA UP, DATA LEFT, DATA RIGHT: move the highlight around the table.

ERASE TABLE: clears all tool information so you can start over. See 11.1.4 below.

JOG: puts the ProtoTRAK VM into the DRO jog operation (see Section 6.3).

RETURN: reverts to the SET UP mode screen.

The electronic handwheels are active, including the fine/coarse selection, while you are in the tool table.

11.1.2 The Logic of the Tool Table

The tool table is organized to do the following:

- Make it easy to set up tools.
- Make it easy to replace a tool or add a tool.
- Retain tool information in memory to reduce set-up.

You assign tool numbers as you write a program. These tool numbers may be from 1 to 99. Before machining, the diameters and Z offset of each of the tools in the program must be defined so that the ProtoTRAK VM can calculate the tool path. Tools that are used in the program that is in current memory are called active tools and their numbers are in red in the tool table.

When you save a program, all the tool information for active tools is saved with it. When the program is opened, the tool information is put into the tool table. This information will replace any information that already is in the tool table for the same tool numbers.

In addition to information about the tools used in a program, you may load in information for tools to be used in 2-axis CNC or in the DRO mode for machining manually. When you tell the ProtoTRAK VM which tool you are using, it will adjust the Z DRO dimensions accordingly so you don't have to touch off and reset after a tool change.

The idea of retaining tool information in memory in order to reduce the amount of set-up needed requires that care be taken to avoid mistakes. Milling work usually requires a lot of tools, many of which are not preset into fixed tool holders. That means tool information that is not very recent is probably no good.

Think of the information in the tool table this way: if you clearly remember setting the tools and entering the diameters very recently, then use the tool table in DRO and CNC run (you can always use TRAKing to remove that last shred of doubt). If you can't remember setting the tools clearly, erase the table and start over – it only takes a moment.

This may cause some confusion because the normal sequence for running a two-axis program is to load in a tool, touch it off and set zero, then press GO. The ProtoTRAK VM will apply the tool offset after the GO press, making the Z dimension meaningless.

You have two choices:

1. Use the tool table, setting the reference and absolute dimension for one of them per the instructions above. This will save you from having to touch off tools every time they are changed in program Run.
2. Don't use the tool table. Erase the entire tool data so that the ProtoTRAK VM will not try to apply offsets.

11.1.3 Initial Tool Set-Up

This procedure is used for setting up tools when the tool table is clear.

1. When you enter this screen for the first time, the words "NOT SET" appear directly under the Z OFFSET column in the REF row. The Data Input Line reads "TOUCHOFF REFERENCE POINT". This is prompting you to establish a reference for the rest of your tools.
2. To establish a reference, put a cutting tool or some other reference setting tool into the spindle and touch the tool to a surface. We recommend that you use something besides a tool that you intend to use machining the job. Ideally, you will have a reference tool that you keep handy for setting up your tools every time. That way, a reference point can be easily re-established later.

We also recommend that you use the top of the vice or table as your reference surface because it is constant and never changes.

3. With the highlight on the screen on the words "NOT SET" and the tool touching some reference point, press SET.

NOTE: If you do use a tool as your reference tool and it breaks, you must retouch off all tools.

4. The words will change from "NOT SET" to "SET" and the highlight will shift to the DIAMETER column of Tool # 1. (Note that you may not be interested in setting up Tool #1 if it is not one of the active tools of the program. If this is the case, use the DATA softkeys to move to a tool you are interested in.)
5. Input the diameter for the tool and press SET.
6. The highlight will move to the Z OFFSET column. Put this tool in the spindle and touch it to the same surface as you used to touch the reference tool in Step 2 above.

7. Press set.
8. The highlight moves to the Z Modifier column. Input a Z modifier if you wish (see below) or simply press SET again for the highlight to move to the DIAMETER column of the next tool.
9. Repeat steps 5 to 8 for each of the tools you want to set up. Remember to touch the same surface you used to set the reference tool.

Once the reference position is set, you will not be able to move the highlight back to the word "SET".

Note: You must set an absolute zero reference in the DRO Mode before machining the part. You may use any tool that you have set up with the above procedure to set your reference, and the ProtoTRAK will automatically compensate for the difference in length for the rest of the tools.

11.1.4 Starting Over: Erasing Tool Information

There will be times when you don't completely trust the information that is in the tool table. For example, perhaps you have loaded in a program that you wrote a month ago and you recall that one of the tools you used was held in a chuck. In that case, you probably want to erase the table and start over.

In order to do this, simply press the ERASE TABLE softkey and answer yes to the prompt. All the data in the tool table will be erased including the reference. The numbers of the tools used in any program in current memory will still be red.

11.1.5 Adding a Tool

When the reference is SET and the original touch-off surface is still available, you can add a tool very easily:

1. First make the tool number active by using it in the program in current memory.
2. Put the new tool in the spindle.
3. Go to the Set-Up Mode, tool table.
4. Enter the diameter.
5. Touch the new tool to the same surface as the reference.
6. Press SET.

If the surface is not available, it will be necessary to establish a new reference before adding the new tool. See Section 11.1.8 below. Once the reference is reset, use the procedure above on the new surface used to set the reference.

11.1.6 Replacing a Tool

If you need to replace a tool that was not used as the reference, simply do the following:

1. Put the replacement tool in the spindle.
2. Put the highlight in the correct row for the tool number.
3. Reenter the diameter if different.
4. Touch the tool to the same surface that was used to touch off the reference.
5. With the highlight in the Z OFFSET column for the correct tool number, press SET.

If you need to replace a tool that was used as a reference, we recommend that you press the ERASE TABLE softkey and start all over again. (Not to nag, but that is why it is a good idea to have a separate reference setting tool and use a constant reference surface. If you work with programs that use a lot of tools, this practice can really save time).

11.1.7 Z Modifiers

Z modifiers make it easy to adjust the depth of cut of particular tools without having to change programmed Z end dimensions or change the tool offsets.

For example, say an end mill was under cutting the depth of a part by .003". An easy way to correct this is to enter a Z modifier.

1. Enter the amount of the adjustment you wish to make. To cut deeper, enter a negative number. To cut shallower, enter a positive number. In the example above, to correct this undercut, we would enter -.003.
2. Press SET.

The ProtoTRAK VM will apply this modifier each time this tool is used.

11.1.8 Resetting the Reference Point

Once the reference reads SET, you are not allowed to highlight and reset it. If you need to reset the reference, there are two ways to change the reference to NOT SET. You can erase the table (and lose all the tool information) or load in a program.

11.1.9 Saving Tool Information

Tool information is saved with the program. If you have made changes to the program or to the tool table that you wish to keep, you must save, or store, the program in the Program In/Out Mode.

11.1.10 Opening a Program

When you open a program, the tool information that is saved with the program will be loaded into the tool table. The numbers for the tools that are used in the program are in red. The diameters, Z Offsets and Z modifiers that were saved with the program will overwrite any information that was in the tool table before the program was opened. If these tools were not set very recently, we recommend that you check them before running the program.

The Ref row will read "NOT SET". A reference may be set at this point.

If you do not go into the tool table after opening a program and before running, you will get a reminder message to check your tools.

11.1.11 Making Tool Set-Ups Easy

We highly recommend the following to make tool set-ups easy.

1. Always use the same tool to set your reference. Preferably, you should use a tool you don't use to machine, something that you keep in your toolbox.
2. Don't use a tool that you use to machine your part as a reference. If your reference tool breaks, you have to reset all your tools.

- Always use the same surface for touching your tools to. Use the machine table, a gage block or the vice, something you can always count on being there. If you use the top of the part, your reference is changing all the time.

11.1.12 The Tool Table and Two-Axis CNC Operation

The information entered in the tool table will also be used when operating the ProtoTRAK VM as a two-axis CNC. Instead of positioning the head, the DRO information seen in the Run Mode will be adjusted for the differences in tools. When a new tool is loaded, the Z dimension will change according to the offsets in the tool table. This change will occur when the GO key is pressed after the "Load Tool # ___" prompt.

11.2 Tool Path

When the TOOL PATH soft key is pressed, the program is processed and the tool path graphics are displayed.

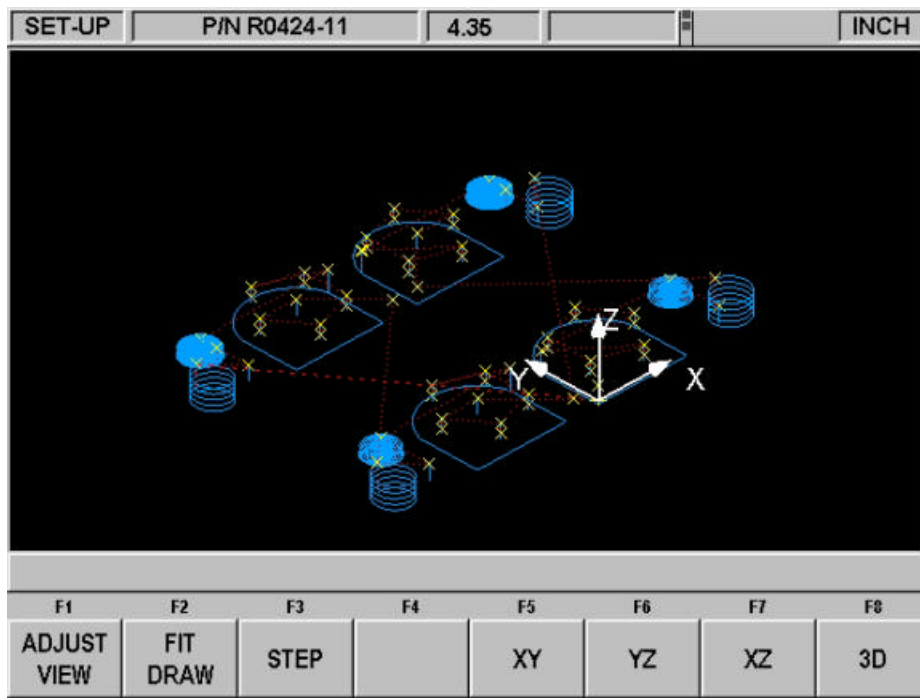


FIGURE 11.2 The Tool Path graphics show the program and tool positions

I01143

Most programming errors that would prevent the program from running are detected when the tool path graphics are selected. For example, if you were to have omitted a minus sign from a Z End dimension, the system would give you an error message that the Z End should not be higher than the Z Rapid.

The displayed graphic is automatically sized to fit the screen and an icon that represents the X, Y and Z orientation is placed at the program's absolute 0 reference point. The path shown on the screen represents the center of the tool.

Colors provide information about tool paths:

- Position and drill events are drawn in yellow.
- Rapid moves are in red.
- Programmed geometry is in blue.

The ProtoTRAK VM will calculate the total amount of time the program will take to run. This total time is displayed in the center of the status line. This total time will not add time for tool changes and pauses. When the program is run, the time displayed is an estimate of the time until the next tool change or the end of the program.

11.2.1 Soft Keys in Tool Path

ADJUST VIEW: calls up additional softkeys to adjust the view. See below.

FIT DRAW: will re-draw, automatically sizing to fit the screen (necessary only if an adjustment changed the drawing from its initial sizing).

STEP: each press of the STEP button shows the next tool move. You may hold the STEP button down to draw the graphic without repeated button presses. To complete the drawing automatically, press FIT DRAW.

XY, YZ, XZ, 3D: shows the same drawing on the screen, with adjustments, in the view you select.

Soft keys in ADJUST VIEW:

FIT: same as the FIT DRAW.

▼ ▲ ◀ ▶ : moves the drawing in that direction.

ZOOM IN, ZOOM OUT: resizes the drawing.

RETURN: returns to the previous soft keys, retaining the adjustments that were made to the drawing.

11.3 Reference Positions (REF POSN)

The Reference Positions screen shows the retract status, the home locations and software limits for all axes.

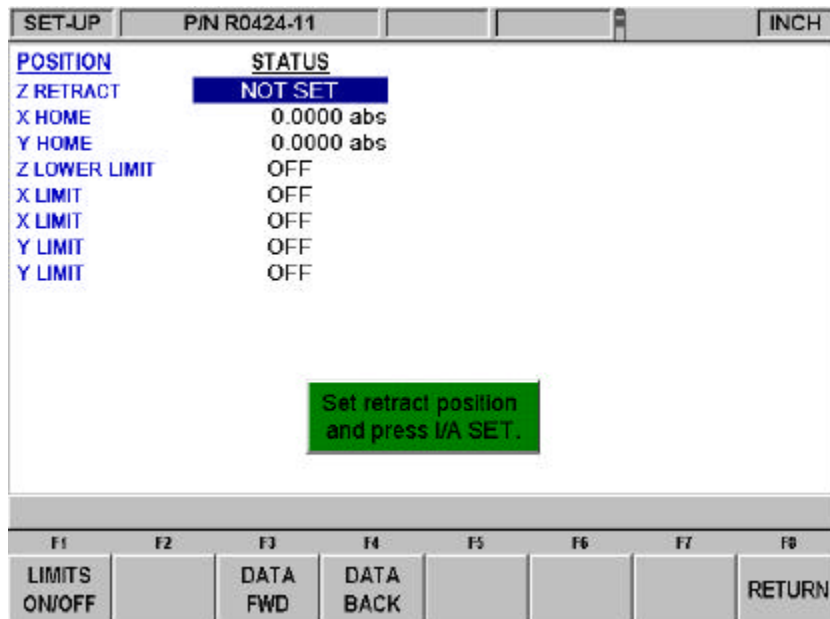


FIGURE 11.3 Reference positions. The Z Retract is not set. Position the head and press a SET key

11.3.1 Z Retract

The Z Retract is where the head will go for a tool change or at the end of running a program. Programs may not be run in three-axis CNC until the Z Retract is set. Since the Z-axis (head) is operated manually in two-axis CNC, it is not necessary to set the Z retract to run a two-axis CNC part.

As a general rule, always set your Z retract so that your longest tool is above the set-up.

When you first enter the Reference Positions screen, the Z Retract will show "NOT SET" and the message window will instruct you to move the ram to the desired retract position and then press SET. You may have to go into the DRO Mode to move the ram to where you want it and then return to the Reference Positions screen to set this position.

11.3.2 Home Positions

X and Y home positions are where the table and saddle go when there is a tool change or at the end of the program. These dimensions must always be from absolute zero. Note Z home is the same as Z Retract. This is only available when you are in 3-axis mode.

11.3.3 Limit Positions

X and Y limit positions (one for plus direction, one for minus) will stop the program if they are exceeded during run. Note that pressing the **LIMIT ON/OFF** soft key will turn the prompted limit off, or back on to its input value. If the limits are turned on, your program and home positions must fit within the limits you define. If you turn on the limits and leave them at the default of 0 Absolute, the program will not run.

11.4 Fixture Offsets

Fixture offsets are entered in the Set-Up Mode. From the screen in Figure 11.0, press the Fix Offset key. The following screen will result.

FIXTURE #	X OFFSET	Y OFFSET	Z OFFSET
1	BASE	BASE	BASE
2	8.1230	0.1160	0.0000
3	0.0000	0.0000	0.0000
4	0.0000	0.0000	0.0000
5	0.0000	0.0000	0.0000
6	0.0000	0.0000	0.0000

X	0.0000	ABS
Y	0.0000	ABS
Z	0.0000	ABS

X OFFSET FROM FIXTURE 1 TO 3 : 0.0000

F1	F2	F3	F4	F5	F6	F7	F8
DATA DOWN	DATA UP	DATA LEFT	DATA RIGHT		ERASE TABLE	JOG	RETURN

Figure 11.4 The Fixture Offset screen.

Setting up fixtures is easy. First, establish your base by setting your X, Y and Z absolutezero. You can do this in the DRO Mode, but the X, Y and Z Absolute position dimensionsare also on this screen for your reference. Fixture #1 is always the base.

Once you set your absolute zero on the base, it is simple a matter of entering the distance from the base to up to five other fixture locations. You can do this one of two ways. By entering the numbers with the keypad or by positioning to the next fixture, putting the cursor on the correct offset value, and then pressing ABS SET.

11.5 Service Codes

These are special codes that may be entered into the ProtoTRAK VM to call up routines used in installation, setting of preferences, machine checkout and service.

The following is a summary of the service codes available. See SWI publication TRAK DPM V Series Installation, Safety and Service Manual (P/N 22632) for detailed information about using service codes.

WARNING!
Before using service codes, be aware that some of the routines are very powerful and may change system settings in a way you don't want. Some of the routines cause the servos to come on and move at rapid speed.

Category	Service Code	Comments
Software version	33	Displays current software versions and system settings.
	37 RS232 Baud rate	To set for RS232 communication
	141 load EEPROM file	To load in set-up values from a disk in the floppy drive
	142 save EEPROM file	To set-up values to a file on a floppy drive
	316 software update – master	Use with floppy disk
	317 software update – slave	
	318 converter activation	
Machine set-up	123 sensor calibration	
	132 handwheel test	
	100 open loop test	DANGER! The machine will move!
	203 brushless motor alignment routine	
	129 arc accuracy setting	To enter the preference.
	301 lube pump cycle time	The frequency of the pumps
	302 lube pump impulse time	The duration of the pumps
	304 toggle X glass scale or TRAK sensor on or off	Run the machine from the motor encoders in case of table scale or sensor failure.
	304 toggle Y glass scale or TRAK sensor on or off	Run the machine from the motor encoders in case of saddle scale or sensor failure.
	323 RS232 com port	Change default com port
Servo calibration	128 backlash calibration constant	
	127 auto backlash configuration	
	11 backlash hysteresis test	
	12 feed forward constant	CAUTION! The servo parameters may change.
Service code number	None	Shortcut to entering the service code when you know it
Lubrication pump switch	Code 300	Runs for programmed time

12.0 RUN MODE

12.1 Run Mode Screen

Press **MODE** and select the **RUN** soft key. The display will show:

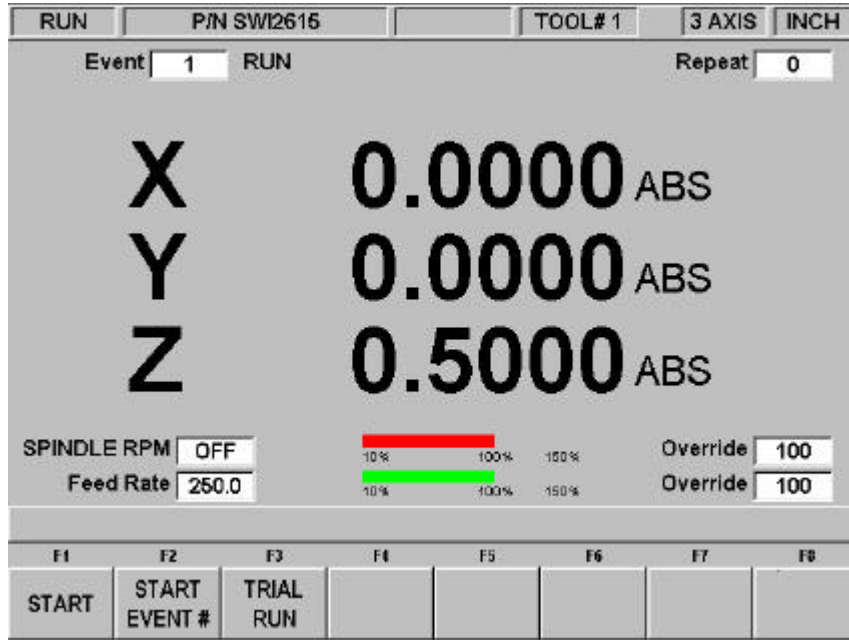


FIGURE 12.1 The Run Mode. The ProtoTRAK VM awaits your instructions for how to begin machining Part Number SWI 2615

Items on the Run Screen:

Event counter: this will be the current event number and event type.

Repeat: if a repeat event is in the event counter, this will show which repeat number, for example, if you program a drill with 5 repeats, this will show which repeat of the event that is being machined.

Spindle RPM: the programmed RPM as adjusted with the spindle override

Red bar: graphical representation of the Spindle override

Override: % of spindle override

Feed Rate: programmed feedrate of the current move as adjusted by the feed override.

Green bar: graphical representation of the feed override

Override: % of feed override.

12.2 Two Versus Three-Axis Running

The three-axis run will control all three axes; the two-axis will control the X and Y (table and saddle) only, with you manually positioning the Z (head).

Most differences that occur as a consequence of either two or three-axis operation are obvious. Two issues are worth noting:

1. The way the tool table works between two and three-axis operation. See Section 11.1
2. The way TRAKing works. See Section 12.5.1

12.3 Starting to Run

Before running a part, you must establish the position relationship between the part and quill. That is, you need to identify where the part is on the table relative to the tool or quill centerline.

This is done by using an edge finder or dial indicator to move the table so that the part program absolute zero is under the quill centerline. **ABS SET** this position as absolute zero in the DRO mode. In addition, load the tool for Event 1 and position it at Z absolute zero. If this is impossible, position the tool some known distance above absolute zero and **ABS SET** this dimension.

The program may be started in the two ways identified as soft keys in the screen in Section 12.1

Pressing the **START** soft key begins the program at Event 1 and assumes that the absolute zero that was last set in the DRO mode corresponds to the part program zero. That is, if you were in the DRO mode and you moved the table to X=0 ABS, and Y=0 ABS the part program zero would be directly under the quill centerline.

Pressing the **START EVNT #** soft key allows you to start in the middle of a program. When you press the **START EVNT #** soft key, the conversation line will prompt "Input Event #." Input the number of the first event you wish to run, and press **SET**. If the **START EVNT #** is a Repeat or Rotate, the conversation line will prompt "Starting Repeat Number" asking which repeat or pass you wish to start.

12.4 Program Run

When you have started by any of the means above, the display will show:

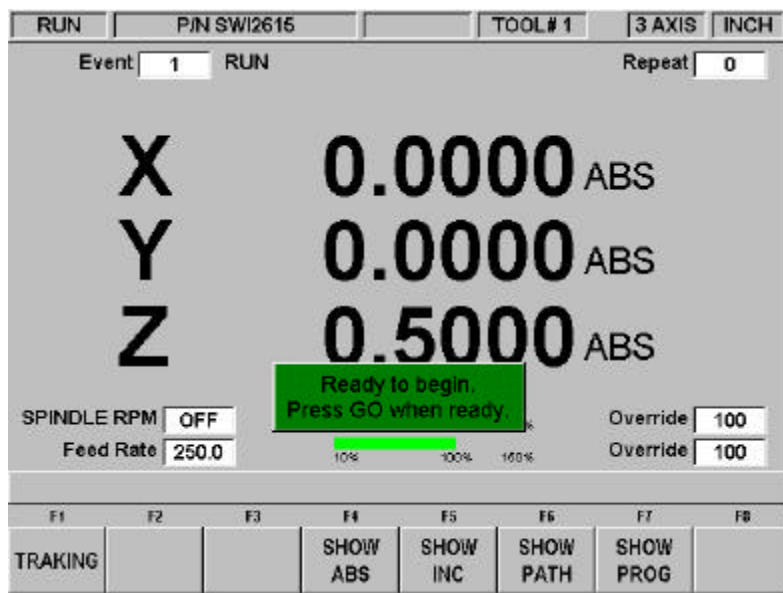


FIGURE 12.4 Either press the GO feed key to start automatic running, or press the TRAKing softkey to control the feed of the CNC

Where:

The part number being run is shown in the status line.

- A "S/F" message will appear in the status line when the Scale Factor is not set at 1.0000
- The event number and type (and the repeat number, if applicable) being run is shown at the top of the screen.
- The current X, Y, Z absolute positions are shown in the information area.
- The **SHOW ABS** soft key (which is automatically assumed if one of the other 3 show keys are not selected) will show the absolute X, Y, Z positions as the part is run.
- The **SHOW INC** soft key will show the incremental (or distance to go within the event) X, Y, Z positions as the part is run.
- The **SHOW PATH** soft key will show the tool path graphics as the part is run.
- The **SHOW PROG** soft key will show the programmed data for the event being run, and the next event as the part is run.

The run procedure is very simple. Follow the instructions on the conversation line and proceed by pressing the GO key.

Once the STOP hard key is pressed, additional softkeys will be available:

TRAKing - select this softkey to control the X, Y and Z programmed motion with the table or saddle handwheel. See Section 12.5 below.

CNC RUN - select this softkey to start the CNC run.

12.5 TRAKing

TRAKing is a special kind of CNC run. When you press the TRAKing softkey, the control of the CNC program head, table and saddle motion is controlled by turning the table or saddle electronic handwheel. Moving the X or Y handwheel in the clockwise direction moves forward through the program; moving counterclockwise moves backward through the program. The spindle must be on in order to TRAK once the head reaches the programmed Z Rapid position. To TRAK slowly, use the Y handwheel. To TRAK quickly, use the X handwheel.

TRAKing comes in handy whenever you are a little unsure about any aspect of your program or set-up. For example, on the first run of a part - instead of pressing GO and holding your hand on the stop button, use TRAKing to bring the tool to the part while you watch the DRO. Once assured that everything is all right, press STOP and get into CNC run.

12.5.1 TRAKing in Two-Axis CNC

When running the ProtoTRAK VM as a two-axis CNC, TRAKing works with the manual operation of the Z or head. The tool may be put in position when the messages "Set Z" or "Check Z" appear. When TRAKing through an XY move, the Z-axis handwheel is not active.

12.6 Program Run Messages

While in the Run Mode, clear instructions and prompts from the VM CNC will tell you exactly what to do to run the program. These messages will appear in a green box in the middle of the screen.

When a tool change is required, the tool information entered in the Tool Table will appear in the green box.

Any Event Comments you entered during programming will appear on the Data Input Line (See section 7.3.2 to use Event Comments).

Once the program starts, a Run Time Clock will appear in the center of the status line at the top of the screen. This clock displays the time remaining until the end of the program or the next tool change, and will count down as the program is run. Note: the program must first be viewed as Tool Path in the Set Up Mode to initialize the Run Time Clock. Otherwise it will show 0.00. (Section 11.2)

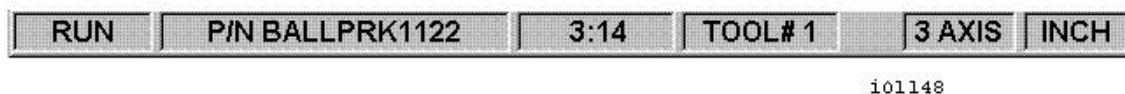


Figure 13.6 The Run Time Clock is in the center of the status line.

12.7 Stop

At any time, the program may be halted by pressing the **STOP** key. This freezes the program at that point. You may choose to continue running the program by pressing the CNC RUN softkey or pressing the **GO** key. You may also run the program by using the table or saddle handwheels by pressing the TRAKing softkey.

12.8 Overrides

The run feedrate may be changed at any time by pressing the **FEED** **↑** or **FEED** **↓** keys. Each press modifies the programmed feedrate, as well as rapid by 10%.

The programmed spindle speed may likewise be changed by pressing the **SPEED** **↑** or **SPEED** **↓** keys.

12.9 Trial Run

Trial Run allows you to quickly check out your program with no Z movement before you actually start to make parts. In trial run the table will move at rapid speed regardless of what feedrates are programmed (the rapid speed may be overridden with **FEED** **↑** and **FEED** **↓** keys). The table will stop at each "stop" location (for example, at each drill location) but immediately continue on without your input.

To do a trial run, press the **TRIAL RUN** soft key from the screen shown in Section 12.1. The message box will read "Ready to begin trial run. Press GO to start." Be certain the table is positioned so that if it moves through the part program, it will not reach its travel limit. Also check that the quill is fully retracted. Press **GO** to begin.

12.10 Data Errors

In order to run, a program must make sense geometrically. For example, you can't machine a .250-inch diameter circular pocket using a .500-inch end mill.

Data errors will nearly always be detected when the ProtoTRAK VM runs through a program--either as a Trial Run or on an actual part run. They may also be detected in the Set Up mode when using the Tool Path Graphics routines.

Whenever the ProtoTRAK VM detects a data error a message will appear that will tell you the error number (you may wish to record this number for troubleshooting purposes) and the event where the error was detected. This is not necessarily the event that is in error since the system often "looks ahead" to make sure there is compatibility from one event to another.

In addition, an explanation is given for each data error type as well as a suggested solution. Press the **RETURN** soft key to go back to the Select Mode screen, correct your error, and proceed.

12.11 Fault Messages

The ProtoTRAK VM performs a number of automatic checks or self-diagnostics. If problems are found a message will appear: "Fault __ __ __ __". The information area will display an explanation and suggested solution.

12.12 Run Sequence

The following sequence of button pushes is the most efficient for running the program.

Mode	To enter Mode screen
Run	To enter Run mode
Start	Either START or START EVENT # plus the number
Go	Takes you back to home, this is needed even if you are there already.
Spindle FWD	Starts spindle. Press spindle only when you finish loading the tool at the "load tool" prompt.
Go or TRAKing	Choose CNC run or TRAKing.

13.0 Program In/Out Mode

From the Select Mode screen, press the PROG IN/OUT softkey. The first screen you see will ask:

"LIST SUPPORTED PROGRAMS ONLY?"

With a highlighted YES or NO.

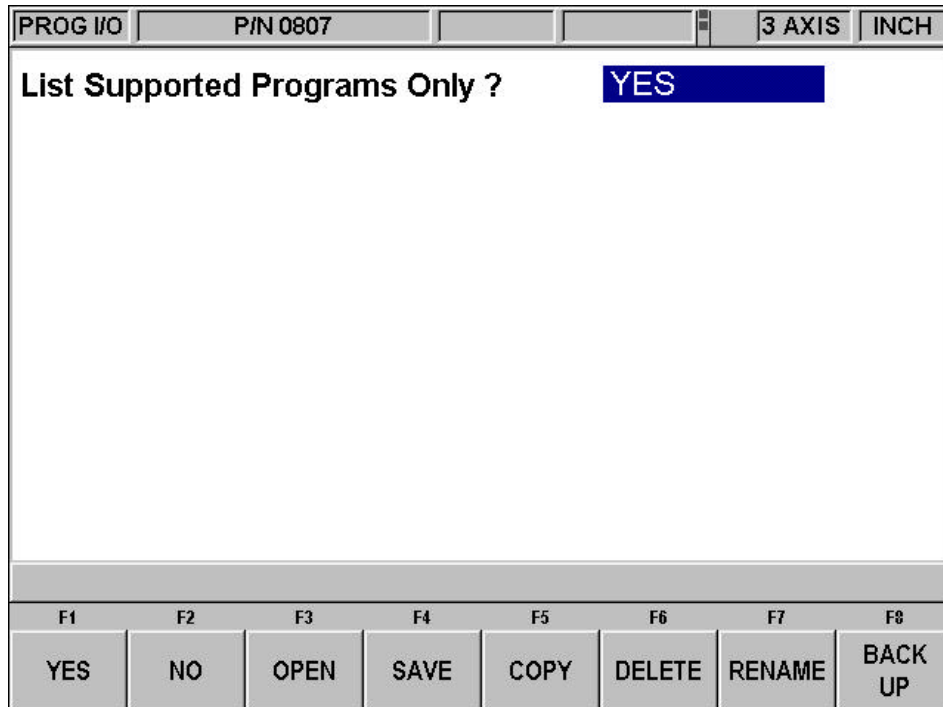


FIGURE 13.0 Supported programs are part programs that can run on the ProtoTRAK VM CNC. You do not have to answer this question every time when you are at this screen. Simply press the softkey for the operation you want.

Supported programs are the programs that will run on your ProtoTRAK VM CNC. It is possible to view other types of files through the Program In/Out Mode, for example, Microsoft Word® files. This type of file is not supported on the ProtoTRAK VM in the sense that you cannot open it and work on it. We recommend a "Yes" response to this prompt. If "No" is selected, it is possible to view, and inadvertently damage, files that are critical to running the ProtoTRAK VM (as well as on other networked computers.)

Filenames and File Extensions

Most places in the ProtoTRAK VM CNC, we refer to the program or part. In Program In/Out Mode, this program or part is called a file. Filenames are program names or part names. They are the name you give to the programs you write on the ProtoTRAK VM, plus a file extension. Although the ProtoTRAK VM can have program names up to 25 characters that use letters and special symbols, most other CNC's must have file names that are eight or fewer characters using numbers only.

File extensions are part of filenames that help describe the file. They appear after the filename and are composed of three letters following a period. For example, *.doc* is the extension that appears after a file name for a file stored using Microsoft Word™.

Usually, but not always, the file name indicates what program was used to create the file. Sometimes this is not the case. Some programs, like those found in early models of CNC, do not attach a file extension to a file name at all. Also, a user may attach his own extension to a file name for his own purposes.

ProtoTRAK and TRAK A.G.E. CNC's always attach an extension to every file that is stored. The extension *.mx2* is used for files, or programs, (written and) stored on a ProtoTRAK MX2, ProtoTRAK M2 or TRAK A.G.E. 2 CNC. The extension *.mx3* is used for the ProtoTRAK MX3, ProtoTRAK M3 and TRAK A.G.E. 3 CNC's. The ProtoTRAK VM uses the extension *.PT4*, whether the program is two or three-axis. (Before opening the file, the ProtoTRAK VM is able to determine what kind of file it is.)

A file extension that is unique to the ProtoTRAK VM is *.GCD*. The *.GCD* extension tells the ProtoTRAK VM that a particular program is a standard RS274, or G Code program. When you specify this extension, the ProtoTRAK VM will treat that program in a special way. This is explained in Section 13.9.2.

13.1 Softkey Selections in the Program In/Out Mode

YES: to display only supported programs.

NO: to display all files.

OPEN: to bring a program from storage into the current memory.

SAVE: to save the program that is in current memory to storage.

COPY: to select and make a copy of a file in storage for pasting in another storage location.

DELETE: to remove a file from a storage location without altering the current memory.

RENAME: to rename a file or folder.

BACK UP: to perform a convenient back up of program files to another storage location.

13.2 Basic Navigation of Program In/Out Mode Screens

The screens in the Program In/Out Mode do not have the normal ProtoTRAK look and feel because they are derived from the Windows operating system. Most functions may be performed using a mouse or keyboard. Softkeys are provided to operate the system through the control's keys.

13.2.1 Basic Parts of the Program In/Out Mode Screens

The status line at the top of the screen will display:

- The mode
- The program name for the program in current memory (if any).
- Whether the ProtoTRAK VM is in two or three axis.

The Look In area shows the storage areas (or drives) and directories that are being displayed below in the listing area.

In the listing area (the biggest part of the screen) appears all the files and folders for the location shown in the Look In box.

The File Name box shows the program file on which the operation will be performed.

Parts of the screen unique to specific operations will be discussed below.

13.2.2 Softkeys in the Program In/Out Mode Screens

Use the softkeys to move around any of the screens in the Program In/Out Mode:

TAB: Moves the highlight between the parts of the screen. Where applicable, tabbing to an area will cause a drop-down box to appear, showing all the selections possible.

DATA FWD, DATA BACK: Moves the highlight up and down through the list. Press and hold for automatic advancement.

OPEN FOLDER: Use this key to open a highlighted folder that contains program files. When the highlight is on the root directory, this will collapse the list displayed and show the next level up. The root directory is represented by a folder with an up arrow followed by two periods. The root directory will disappear when the most basic organization for the drive in the Look In box is reached.

13.3 Opening a File

To open a program file from a storage location, press the OPEN softkey from the Program In/Out Mode screen. The ProtoTRAK VM will always default to the last folder you had open.

Find the file using the softkeys as described above in the section on basic navigation.

In addition to the basic parts of the screen described above, two additional parts of the screen appear in the open operation:

File Name: - Displays the name of the file that is highlighted from the list.

Open As: - lists the formats for which the file may be opened. The default is .PT4.

Two additional softkeys appear:

OPEN FILE: Opens the highlighted program file and puts it in current memory. Only one file may be in current memory at a time, if one is there already, a warning message will appear before that file is overwritten.

RETURN: Returns to the Program In/Out Mode screen.

When the open operation is finished, the system will return to the Select Mode screen.

13.3.1 Preview Graphics

As an aid to finding the file you want to open, the ProtoTRAK VM allows you to look at the part graphics before opening. Simply select the file and press the LOOK hardkey. The screen will show the part graphics. Press LOOK again or Return to revert back to the Program In/Out Screen.

The graphics displayed in this process are not exact, but are a handy representation of the program.

Note: DXF and GCD files may not be previewed with the LOOK feature.

13.4 Saving Programs

To save a program file to a storage location, press the SAVE softkey from the Program In/Out Mode screen.

Find the drive and folder you want to save the program file in using the softkeys as described above in the section on basic navigation.

Three additional parts of the screen appear once the SAVE softkey is pressed:

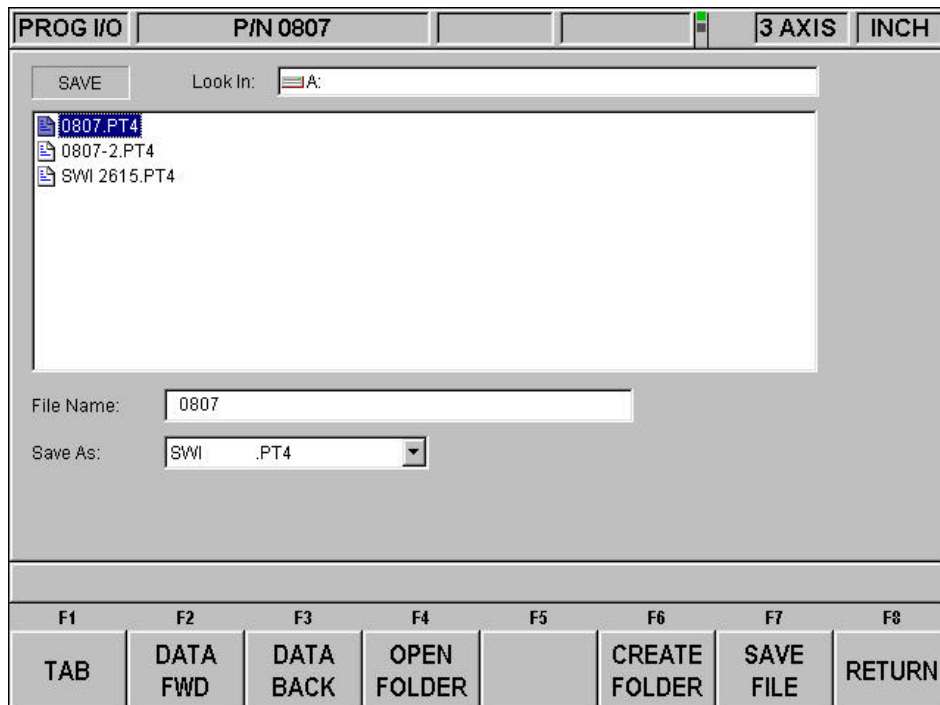


FIGURE 13.4 The Save screen

File Name: displays the name of the file that is in current memory.

Save As: lists the formats for which the file may be saved. The default is .PT4.

Three additional softkeys appear:

CREATE FOLDER: Use this to create a new folder for the program file. This new folder will be added to the list shown in the listing area, at the same level of organization as the files and folders shown. Once the CREATE FOLDER softkey is pressed, a Data Input Line will appear for entering the name of the folder. The name "Folder1" will be written in the box. To accept this name, press SET. You may input a name you select by writing over this name. Use the same procedure for naming a program (see Section 7.3.1).

SAVE FILE: Saves the program file to the location shown in the Look In area.

RETURN: Returns to the Program In/Out Mode screen.

Once the save operation is finished, you will see the file name added to the files in the listing area.

13.5 Copying Programs

To copy a program file from one storage location to another, press the COPY softkey from the Program In/Out Mode screen. Only one file may be copied at a time using this operation. To copy multiple files or folders, see Section 13.8 Back up.

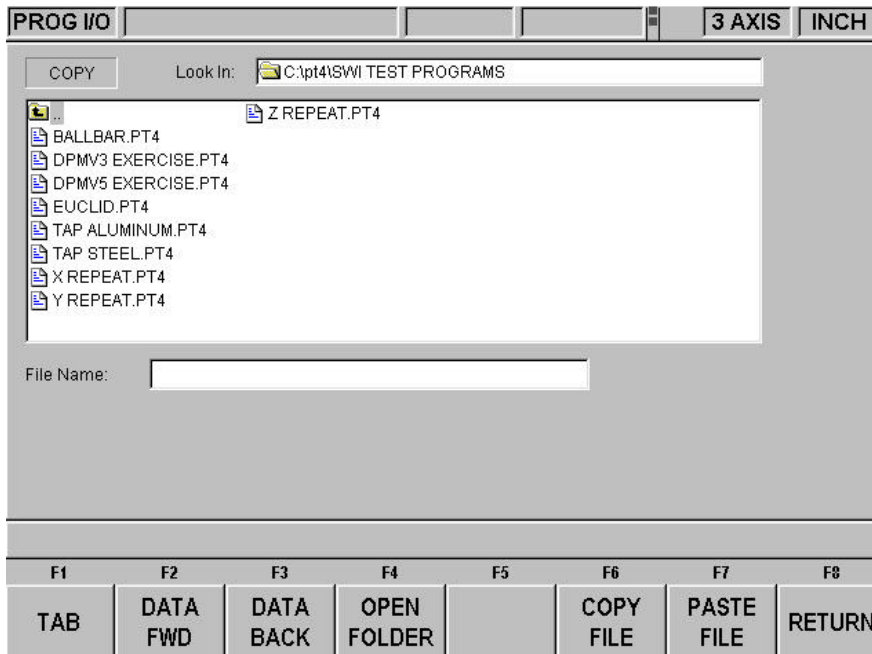


FIGURE 13.5 *The Copy screen*

The copy operation is in two parts. First, use the navigation procedure described in Section 13.2 above and highlight the program you wish to copy. Press the COPY FILE softkey to copy the file. Then go to the new file or drive, open it using the Open Folder softkey and press Paste file. Once the file has been copied, it can be pasted to as many other locations as you want.

Additional softkeys in COPY:

COPY FILE: Makes a copy of the highlighted file.

PASTE FILE: Writes a copy of the file to the location shown in the Look In box.

RETURN: Returns to the Program In/Out Mode screen.

When the pasting operation is finished, you will see the file name added to the listing area.

13.6 Deleting Programs

Programs in current memory are removed from current memory in Edit Mode. See Section 10.3

To remove a program file from a storage location, press the DELETE softkey from the Program In/Out Mode screen.

Use the navigation procedure described in Section 13.2 above and highlight the program file or folder you wish to delete. Press the DELETE FILE or DELETE FOLDER softkey. A warning message will appear for confirmation.

Additional Softkeys in DELETE:

DELELE FILE: Press this to delete a file.

DELETE FOLDER: Press this to delete a folder.

Softkeys that appear with the confirmation message:

YES: Press this if you want to delete.

NO: Press this if you do not want to delete. The delete operation will be aborted and the previous softkey selections will return.

When the delete operation is finished, the file or folder name will disappear from the listing area.

13.7 Renaming

To rename either a file or a folder, press the RENAME softkey from the Program In/Out Mode screen.

To rename a file or folder:

1. Use the navigation procedure described in Section 13.2 above and highlight the program file or folder you wish to rename.
2. TAB to the New Name area and enter a new name. Use the same procedure as for naming a program (see Section 7.3.1).
3. TAB to the New Extension and enter a new extension.
4. Press either RENAME FILE or RENAME FOLDER.

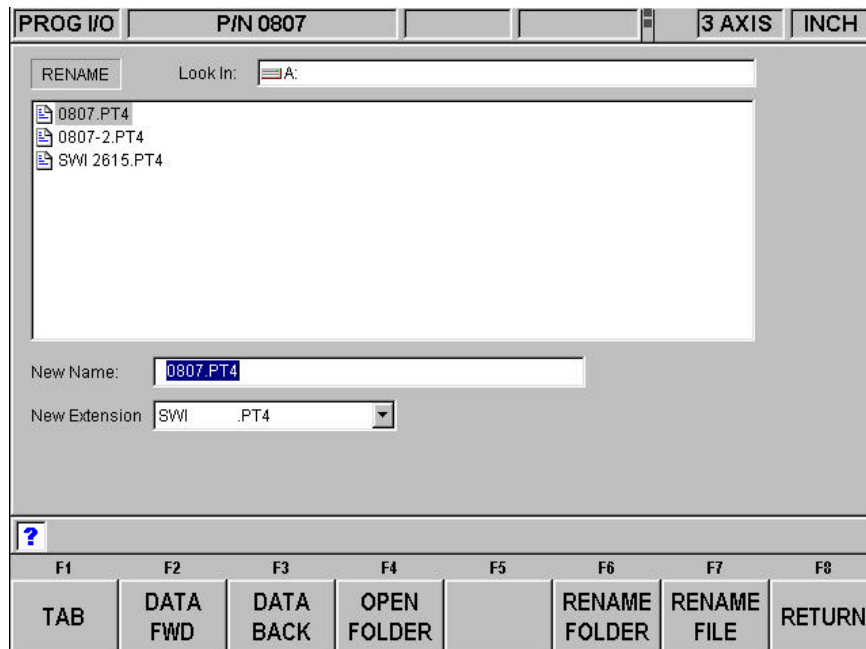


FIGURE 13.7 Renaming a file. Press the Help hard key to call up the alpha keys

Additional parts of the screen appear once the RENAME softkey is pressed:

New Name: When a file or folder is highlighted, the name will appear here. When the TAB, the RENAME FILE or RENAME FOLDER softkey is pressed, the highlight will move here and you will then be able to write in a new name.

New Extension: A new extension can be given to the file picking from the ones available. If the file name already contains an extension, you will have to erase the old one before giving it a new one.

Additional softkeys:

RENAME FOLDER - Press once a new name has been entered into the New Name and New Extension areas to change the name of the folder.

RENAME FILE- Press once a new name has been entered into the New Name and New Extension areas to change the name of the file.

RETURN - Returns to the Program In/Out Mode screen.

13.8 Backing Up

In order to protect your important programs, it is a good idea to back them up regularly. That way, if a floppy disk or hard drive becomes unusable, you will not have to re-write the program.

To back up your files, press the BACK UP softkey from the Program In/Out Mode screen.

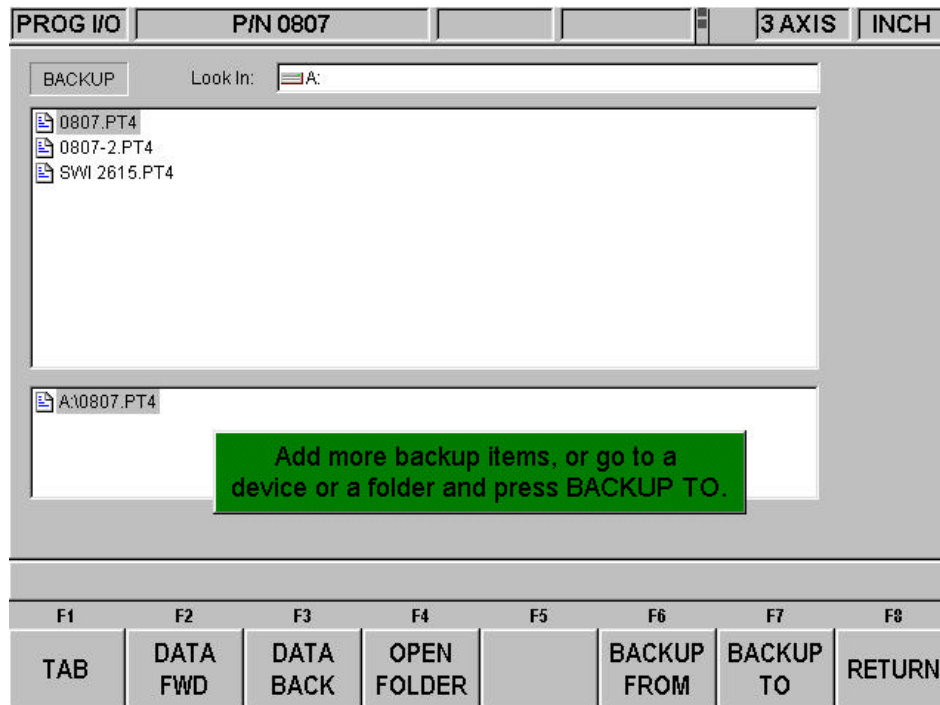


FIGURE 13.8 Backing up. The top part of the screen shows all the items in Drive A. The bottom part shows the items that have been picked for backing up

The basic procedure for backing up is:

1. Use the navigation procedure described in Section 13.2 above and highlight the program file or folder you wish to back up.
2. Press the BACKUP FROM softkey. You will see the item appear, along with its directory path, in the new listing area under the main listing area.
3. Repeat the above for as many items as you wish.
4. Use the navigation procedure to select a different drive or a different folder.
5. Open the drive or folder using the Open folder key.
6. Press BACKUP TO.

When the back up operation is completed, you will see the items and their directories in the new location.

Note: it is good practice to back up files to a different drive, rather than to a different folder on the same drive. For example, if you keep your programs on the ProtoTRAK VM hard drive, it is a good idea to back them up on a floppy disk or to another computer that is networked into the ProtoTRAK VM. That way, if the ProtoTRAK VM hard drive becomes unusable, you will have the part programs somewhere else so that you can reload them when the problem with the ProtoTRAK VM hard drive is resolved.

13.9 Converters™

Converters are programs within the ProtoTRAK VM that translate CNC program files of another format into a ProtoTRAK VM file, or a ProtoTRAK VM file into a different format. With converters, you can run programs written on the ProtoTRAK VM on a machine that does not have a ProtoTRAK VM CNC and vice versa.

Each ProtoTRAK VM comes with converters for other ProtoTRAK and TRAK CNC's. Converters for other brands of CNC's are sold separately.

Program conversions take place by first translating the file into a neutral run engine, then from neutral to the desired file format. For this reason, you should think of conversions as being only one way. The conversion process changes the file in ways that are harmless and so the results are correct. However, when converted back, it will not be the same as it was originally written; it will create the same part, but some of the lines of code will be different.

13.9.1 Activating Converters

Converters must be activated before you can use them. Standard converters include those that handle the translation between the ProtoTRAK VM and other TRAK CNC's. Optional converters are purchased separately. Standard converters and optional converters that are ordered and shipped with the machine are activated at the factory.

You can tell which converters are activated by looking in the Open As (see Figure 13.9.3) or Save As windows (see Figure 13.4).

If you purchase a converter after you have installed your machine, you must activate it yourself using a simple procedure:

1. Go to Set Up Mode, Service Codes, then A. Software.
2. Press the soft key for Service Code 318.

3. A screen will appear that lists all the converters available for the version of ProtoTRAK VM software that you have. The converters that are active are listed in black; the inactive converters are grayed out. See Figure 13.9.1.
4. Place the cursor over the inactive converter you want.
5. Press the Install soft key.
6. There is a password that enables you to use the converter. Enter the password in the Data Input Line and press Set. If you do not have the password, call the SWI Customer Service Group at (800) 367-3165. You will need to make arrangements to pay the fee for the converter before you are given the password.

1.	Southwestern Industries, Inc.	SWI	.MX2
2.	Southwestern Industries, Inc.	SWI	.MX3
3.	Southwestern Industries, Inc. (F6M)	F6M	.CAM
4.	Haas Automation, Inc.	HAAS	.HS1
5.	Southwestern Industries, Inc. (GCD)	SWI	.GCD
6.	AutoDesk, Inc. (AutoCad DXF)	ACAD	.DXF

F1	F2	F3	F4	F5	F6	F7	F8
PAGE FWD	PAGE BACK	DATA FWD	DATA BACK		INSTALL		RETURN

FIGURE 13.9.1 Service Code 318 displays a list of converters available on the ProtoTRAK VM software version you are running. On this list, the first three converters are active, the rest are not active.

13.9.2 Converting From a Different Format into a ProtoTRAK VM

Conversions from a different format into a ProtoTRAK VM occur when the file is opened.

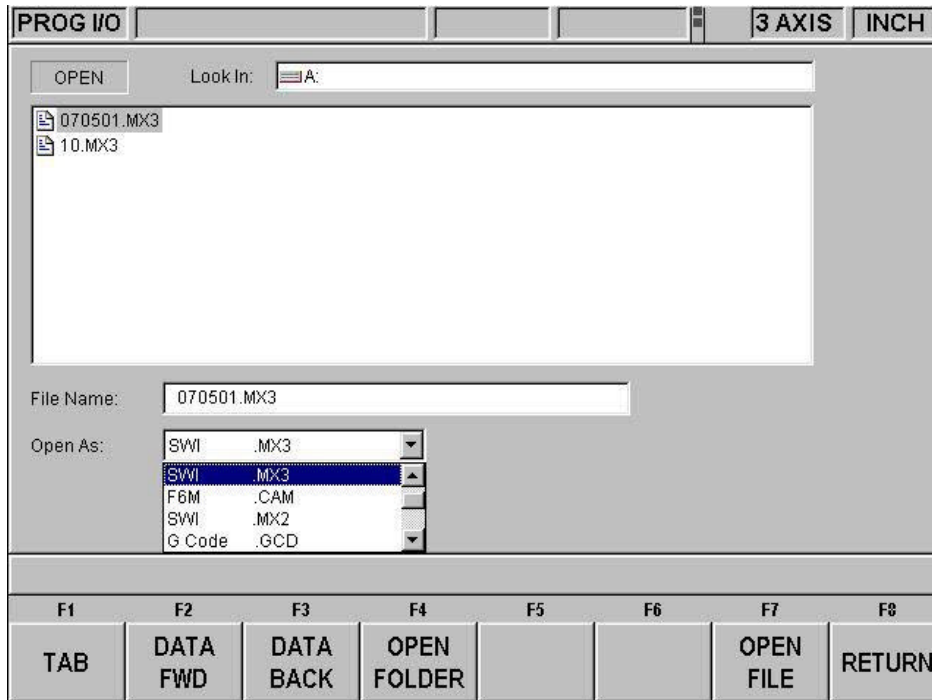


FIGURE 13.9.2 Use the Open As box to tell the ProtoTRAK VM what kind of file it is

Use the Open As box to tell the ProtoTRAK VM what format the file is in so that it knows how to convert it to the ProtoTRAK VM format. In Figure 13.9.1 the ProtoTRAK VM could guess that the file to be converted was from a previous version of ProtoTRAK because of its file extension (.mx3). But since file extensions may be missing or may not really describe the file format correctly, you can use the Open As box to declare the file type.

All files, or programs, open on the ProtoTRAK VM as a .PT4 file (with one exception, G-Code files, see below). Once the file is opened as ProtoTRAK VM file, you may store it as ProtoTRAK VM file with the same filename and the extension .PT4.

The drop-down menu in the Open As box shows which converters are available. Open As types that are grayed out indicate converters that are available for purchase.

13.9.3 Opening G-Code Files

The one Open As file type that the ProtoTRAK VM does not convert into its own format are files that you specify as G Code .GCD files. You can use this designation to tell the ProtoTRAK VM that you want this file to run directly, without it converting. You would do this if you had a very large CAM file. The ProtoTRAK VM can handle the G code more efficiently when the program is very large because the ProtoTRAK VM format is set up to handle complex geometry and CAM files are large because they break everything down to small XYZ position moves.

When the .GCD designation is used and a file is opened and not converted, you will have limited access to the program within the ProtoTRAK VM operating system. You may view

the tool path and run the program in Run Mode, but you will not be able to edit or view the program in Program mode.

C-Code Files may be editing in the Edit Mode.

13.9.4 Converting From the ProtoTRAK VM to a Different Format

Files, or programs, are converted from the ProtoTRAK VM to a different format using the Save function of Program In/Out Mode.

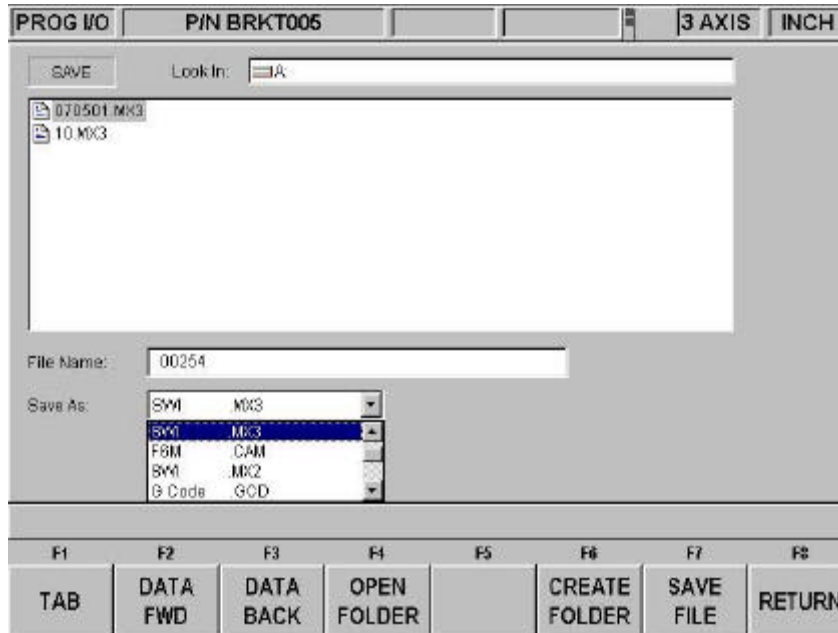


FIGURE 13.9.3 Use the Save As box to tell the ProtoTRAK VM what file format you want to end up with

Use the Save As box to tell the ProtoTRAK VM what kind of file you want the current program (in the .PT4 format) to be converted into.

In Figure 13.9.3 the file name 00254 is being saved on drive A as a .mx3 file. Note that although the program or part name as shown in the status line is BRKT005, the file name given for converting the file conforms to the .mx3 format – fewer than eight characters long and consisting of numbers.

13.10 ProtoTRAK and TRAK CNC Compatibility

File exchange between the ProtoTRAK VM CNC and other ProtoTRAK and TRAK CNCs is possible because the ProtoTRAK VM CNC is backward compatible. In other words, the ProtoTRAK VM CNC can store and retrieve other ProtoTRAK and TRAK CNC files. The actual transfer of the files can be accomplished by using a floppy disk, zip disk, RS232 and/or Ethernet cable. In order to transfer files between the ProtoTRAK VM CNC and previous generations of ProtoTRAK and TRAK CNC, you must have the .MX2 and .MX3 converters activated. See Section 13.9 above.

Note: Previous ProtoTRAK and TRAK CNCs allow numeric filenames of eight (8) characters or less, while the ProtoTRAK VM CNC allow alphanumeric filenames (letters and numbers) of up to twenty-five (25) characters. Be sure to use only numeric filenames when storing a file on a ProtoTRAK VM CNC that will be retrieved by previous

ProtoTRAK and TRAK CNCs. Before conversion, you can easily rename the file in the ProtoTRAK VM CNC current memory.

13.10.1 File Formats

The ProtoTRAK VM CNC can store and retrieve the following ProtoTRAK and TRAK CNC "dot" file formats.

Previous ProtoTRAK and TRAK CNC "dot" file formats:

ProtoTRAK M2	.mx2 (pronounced "dot" mx2)
ProtoTRAK MX2	.mx2
ProtoTRAK MX2E	.mx2
TRAK AGE2	.mx2
ProtoTRAK EDGE	.mx2
ProtoTRAK M3	.mx3
ProtoTRAK MX3	.mx3
ProtoTRAK MX3E	.mx3
TRAK AGE3	.mx3
TRAK QMV	.mx3

13.10.2 Opening .MX2 and .MX3 files on a ProtoTRAK VM CNC

Programs written on previous generation ProtoTRAK and TRAK CNCs may be opened and run on the ProtoTRAK VM CNC. You will need to have the .MX2 or MX3 converters activated (see Section 13.9 above). The ProtoTRAK VM CNC will automatically convert the file (.MX2 or .MX3) to a .PT4 file. The original file will remain on the storage device unchanged and the converted file will be in current memory. You will have to save the converted file using the procedure in Section 13.4 above in order to place in into storage.

Since there are some feature differences between the ProtoTRAK VM and previous generation controls, you will have to pay attention to the following:

Event or feature	Comment	Result
Nested Repeats	These are handled a bit differently between the controls. The Nested Repeat must reference all event numbers from a previous Repeat event, not just the previous Repeat event number, otherwise it will cause an error message to appear when the event is run.	For example Not Acceptable: Event #13 Repeat Events #1-12; Event #14 Repeat Events #13. Acceptable: Event #13 Repeat Events #1-12; Event #14 Repeat Events #1-13.
3 or 4 Sided Pocket	These Events will be recognized by the ProtoTRAK VM CNC but in a modified format.	The ProtoTRAK VM CNC will convert the 3 or 4 Sided Pocket routines into an Irregular Pocket Event.
RPM	For TRAK and ProtoTRAK CNCs where RPM is not programmed, the RPM on the ProtoTRAK VM will be left blank.	We recommend that you inspect the program in program mode or search edit before running in order to adjust or add the RPM.

13.10.3 Running ProtoTRAK VM Files on ProtoTRAK and TRAK CNC Controls

In order to have a program written on a ProtoTRAK VM run on a previous version ProtoTRAK or TRAK CNC, you will need the .MX2 and .MX3 converters activated (see Section 13.9 above). Save the program as either a .MX2 or .MX3 file (depending on the control or program you want to run).

Since there are some feature differences between the CNCs the process will generally yield a useable .mx2 or .mx3 program but with the following exceptions:

Event or feature	Comment	Result
Hidden Areas in Irregular Pocket	The ProtoTRAK or TRAK CNC does not recognize Hidden Areas in Irregular Pockets.	The Irregular Pocket will be converted to an Irregular Pocket, however, the ProtoTRAK or TRAK CNC will display an error message that there are Hidden Areas in the Irregular Pocket. We recommend that you separate the Irregular Pocket into two or more Irregular Pockets using the ProtoTRAK VM before conversion.
Tap Events	This routine does not exist many versions of the ProtoTRAK or TRAK CNC.	The routine will be ignored in the converted program. We recommend that you reprogram the Tap Events into Drill or Position events before conversion.
Subroutines with %Feed or %RPM	The %Feed or %RPM function does not exist on the ProtoTRAK or TRAK CNC.	The %Feed or %RPM information will be removed from the Subroutines. The programmed feed rates will be run. We recommend that you inspect the feed rates before running the program on the ProtoTRAK or TRAK CNC when the %s are other than 100%.
Tool Path Programming	Only Part Geometry programming is supported on the ProtoTRAK or TRAK CNC.	You can only transfer Part Geometry programs to the ProtoTRAK or TRAK CNC.
Zig-Zag Entry Mode Ramps	This routine does not exist in the ProtoTRAK or TRAK CNC.	The routine will be converted to a Plunge routine. We recommend that you check your Z feedrate to make sure it will be correct for a plunge.
Event Comments	Event comments are not supported on the ProtoTRAK or TRAK CNC.	Event comments will be ignored.
Thread Mill	This routine does not exist in the ProtoTRAK or TRAK CNC.	Thread Mill events will be ignored. We recommend that you replace these events with the Helix Events and Mill Events to ramp in and ramp out of the helix.
Tool Table Info	The Z offset information cannot be read into the previous generation ProtoTRAK or TRAK CNC, just the tool number and diameter.	The Z-axis tool table information will have to be set in the ProtoTRAK or TRAK CNC.

Event or feature	Comment	Result
Event or feature	The ProtoTRAK or TRAK CNC does not contain an Irregular Profile Event.	The Irregular Profile Event will be converted to Mill and Arc Events and the programming of the finish cut and passes will be lost. We recommend that after conversion, you add repeat events for the steps and finish cut, using the technique of overstating the size of the cutter you will use to cut the profile on the roughing passes.

13.11 Running G Code Files

The ProtoTRAK VM allows you to run G Code files directly without having them converted to the ProtoTRAK VM programming format. You may want to do this if you have a very large CAM file made up of small XYZ position moves, or if there is complex surface contouring. In these cases, the ProtoTRAK VM can handle the files more efficiently by running the G Code directly. While running the G Code file directly does not give you the benefit of the easy programming format of the ProtoTRAK VM, you are not likely to be able to use this benefit with a very large or complex file anyway.

To run the G Code file directly, open the file using OPEN AS: G Code .GCD. The entire program will be brought into current memory. You will be able to view the tool path when you run the program in the Run Mode, but you will not be able to edit the program or view it in the Program Mode. In order to edit the program, you will have to use the same application (usually CAM) that you used to write the program originally.

You may edit G Code Files in the Edit Mode.

13.11.1 G Codes Recognized by the ProtoTRAK VM

G00	positioning (rapid)
G01	linear interpolation (feed)
G02	circular interpolation CW
G03	circular interpolation CCW
G06	CW Helix
G07	CCW Helix
G16	Selects a vertical plane via a bearing angle (value in 'D' word) from the x axis
G17	Selects the XY plane for circular interpolation.
G18	Selects the XZ plane for circular interpolation.
G19	Selects the YZ plane for circular interpolation.
G20	input in inch
G21	input in mm
G40	cutter compensation cancel (for SWI it means center)
G41	cutter compensation left
G42	cutter compensation right
G61	exact stop check mode
G64	cutting mode (no hesitation between events. NOHES=true)
G80	Hole machining canned cycle
G81	Drill canned cycle
G82	Spot drilling canned cycle
G83	Peck drilling canned cycle
G84	Tapping canned cycle
G85	Boring canned cycle

13.11.2 M Codes Supported by the ProtoTRAK VM

M00	program stop with prompt (press go to procd.)
M01	optional stop
M02	end of program (no rewind)
M03	spindle CW
M04	spindle CCW
M05	spindle stop
M06	tool change
M07	mist coolant ON
M08	flood coolant ON
M09	coolant OFF
M30	end program (rewind stop)
M79	Send SWI 'O' (ascii 79) commands, value in 'P' word
M98	Subroutine Call to block (PWORD), repeat (L WORD)

13.11.3 Valid Characters for Word/Address Sequences

G	Prepare to execute a G COMMAND	ParseGcode
M	Prepare to execute a M COMMAND	ParseMcode
N	Introduces a block number	ParseEventNum
T	Specifies the tool number to use	ParseToolNum
F	Specifies a feedrate	ParseFcode
S	Specifies a spindle rpm	ParseScode
D	Specifies the diameter for the current tool	ParseDval,
E	Optional parameter	ParseEval,
X	Specifies the X dimension	ParseXval,
Y	Specifies the Y dimension	ParseYval,
Z	Specifies the Z dimension	ParseZval,
I	Specifies the incremental X dimension	ParseIval
J	Specifies the incremental Y dimension	ParseJval
K	Specifies the incremental Z dimension	ParseKval
L	An Optional Parameter	ParseLval
P	An Optional Parameter	ParsePval
(Introduces a comment	ParseComment

13.12 Networking

The subject of networking is extensive. This portion of the manual will give you basic instructions for setting up a simple peer-to-peer network and some system information useful to network administrators.

A network is simply two or more computers connected (usually by a cable) so they may share information. Networks within a single building are called LANs for Local Area Network.

The benefit of networking is that you can move information easily between computers. This ease of use enables some handy functionality, for example:

1. An effective file Back-up routine. File back-ups are essential if you want to retain programs for future use. Any hard drive or floppy drive could fail. Having program files backed up to a different location saves you from rewriting the programs from scratch if a failure occurs.
2. An easy way to import CAD/CAM or DXF files from other computers.

3. An effective revision control. Having a single shared folder on the network enables you to have a single location with all the latest versions of the programs.

Of course, the above functions are possible without a network by shuttling floppy disks around. The reason to have a network is that it saves time. Once it is set up, you can do repetitive functions without much work. For example, if a particular job requires you to run a CAM file that you don't have on the ProtoTRAK VM already, going to the pre-arranged networked location using the Program In/Out of the VM gets you going right away. Without networking, someone has to make you a disk with the file on it. Another example is program file back-ups. With networking, you can back-up with a simple routine in the Program In/Out Mode. Without networking, you must have a good system for managing floppy disks, including labeling, storage and retrieving program files. You are more likely to do regular back ups if the process is easier.

Networking can be tricky. If you do not have experience setting up a network, be warned. Computer companies haven't done for networks what we have done for CNCs. Getting everything to work properly can require hours of troubleshooting, even for experts. There are instructions below to guide you through the most basic case of establishing a peer-to-peer network. Beyond that, you should consult a qualified Network Administrator.

13.12.1 Network Security

Network security with the ProtoTRAK VM is very straightforward. Within the context of a network, the ProtoTRAK should be regarded more as a device than as a computer, that is, it is more like a printer hooked to the network than a desktop PC. The "programs" it generates take the form of simple text files that have a very small security risk. When operated as a ProtoTRAK, it cannot generate .exe or other files. It can only generate .txt files. It is possible for a virus to enter the ProtoTRAK and do damage, but it is very unlikely. If that does happen, you can send it to us for repair or replacement under our well-established product support systems. To eliminate the (miniscule) security risk even further, a server or router may be inserted in the network between the ProtoTRAK and the rest of the network.

13.12.2 Peer-to-Peer Networking

The most basic type of networking is called *peer-to-peer networking*. This involves simply connecting two or more computers together with a cable attached to a *network interface card*, which is installed into each computer.

Network Interface Card (also known as an Ethernet Card): an electronic circuit adapter card that is placed inside a computer and allows it to attach to a special network cable.

Network Cable: The network cable is what physically plugs into a network interface card to allow the computers to communicate with one another.

There are many types of network cables. The network interface card inside the ProtoTRAK VM CNC control requires a *twisted-pair category 5 rated network cable*, which looks like a telephone cable and is readily available at most computer supply stores.

In a peer-to-peer network, all the computers that are connected together are thought of as peers, or equals. The ProtoTRAK VM CNC control can be considered a peer to other CNC's or desktop computers.

Peer-to-peer networking features are built right into the Windows 95/98/ME/XP software. If you are using any of these Microsoft operating systems you do not have to purchase

any additional software to set up this type of network. The ProtoTRAK VM CNC control uses the Windows 98 2nd Edition operating system.

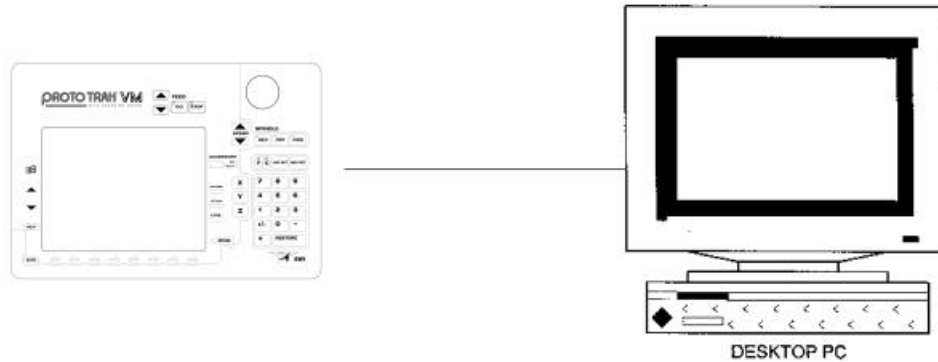


FIGURE 13.12.2 Peer-to-Peer (desktop PC to ProtoTRAK)

13.12.3 Basic Network Set-Up

This section will give you instructions for the most basic set-up of one ProtoTRAK with one desktop computer. The desktop computer must have an Ethernet card installed. Use a crossover CAT5 cable to connect the machines. Before you begin, connect a keyboard and mouse to your ProtoTRAK and turn the ProtoTRAK on with them connected so that it will recognize them. If your set-up is more complicated than that pictured in Figure 13.12.2, you should skip this section and see Section 13.12.4.

1. For the ProtoTRAK VL and VM CNCs, press the Windows key and then press Windows 98. For the ProtoTRAK SM and VM, press the SYS key, then press CONFIG NETWORK. This will take you to the Windows desktop.
2. From there, click on START, then go to SETTINGS, CONTROL PANEL.
3. Double click the "NETWORK" icon, and go to the Configuration tab. You should get a window that looks something like the one below.

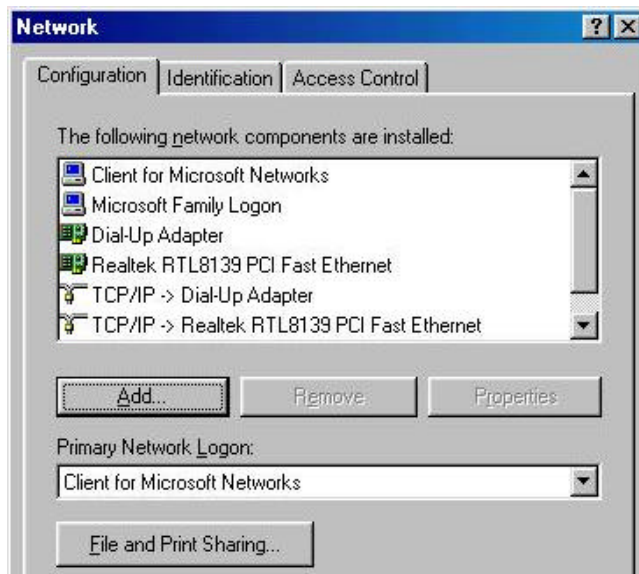


Figure 13.12.3.1

There are 4 components that are needed:

- Client for Microsoft Networks
- Realtek PCI Fast Ethernet Adapter
- TCP/IP
- File and Printer sharing.

4. Double click on Realtek PCI Fast Ethernet, it should be the one with the green icon next to it.
5. Under the Bindings tab, make sure that you have the option checked off which should read TCP/IP as shown in Figure 13.12.3.2.

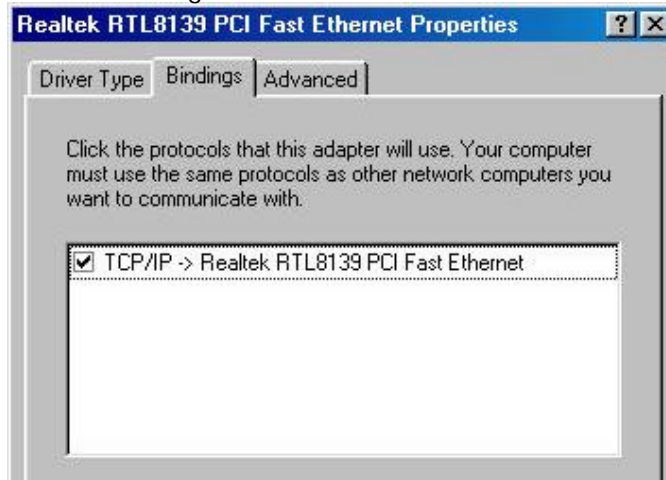


Figure 13.12.3.2

6. Exit this window and go back to the first Network window.
7. Highlight the line which reads "TCP/IP—Realtek PCI Fast Ethernet"
8. Click on "Properties", under the "IP address" tab, select "Specify an IP address". Now type in an IP address, making sure to match the first set of numbers with the IP address of the computer that you are connecting to. (You can obtain your computer's current IP number by going to START -> RUN -> WINIPCFG. Otherwise you can manually enter one).

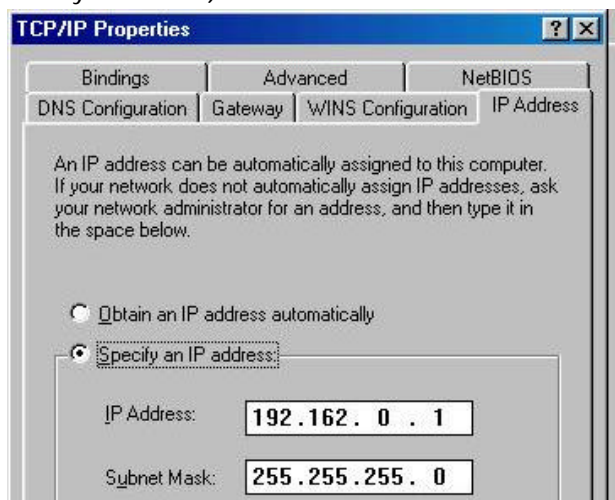


Figure 13.12.3.3

9. Click on the "DNS Configuration" tab, and make sure that it is disabled.

10. Click on the “WINS Configuration” tab, and make sure that is disabled as well.
11. Click on the “Bindings” tab, and make sure that you have “Client for Microsoft Network” and “File and printer sharing” checked. You will need these in order to log on correctly as well as be able to transfer files from one computer to the other.

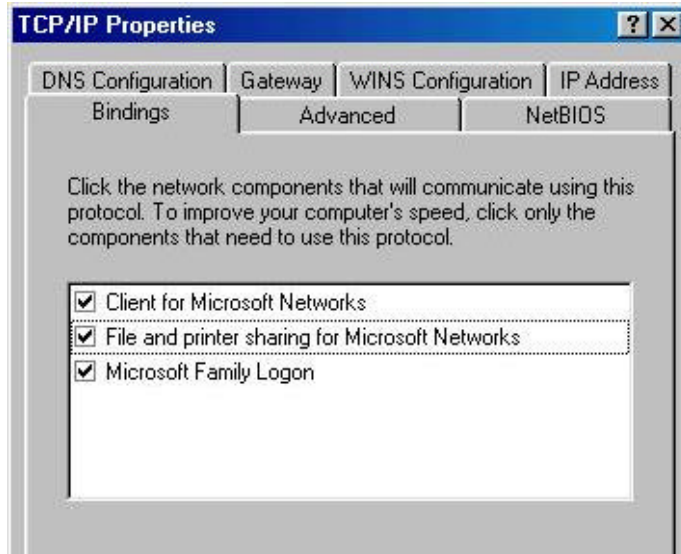


Figure 13.12.3.4 TCP bindings

12. Close this window and return to the original Network window. Click on the “Identification” tab.
13. Compare these settings to the ones on your computer, and make sure that workgroup is the same for both, and that they each have unique computer names.

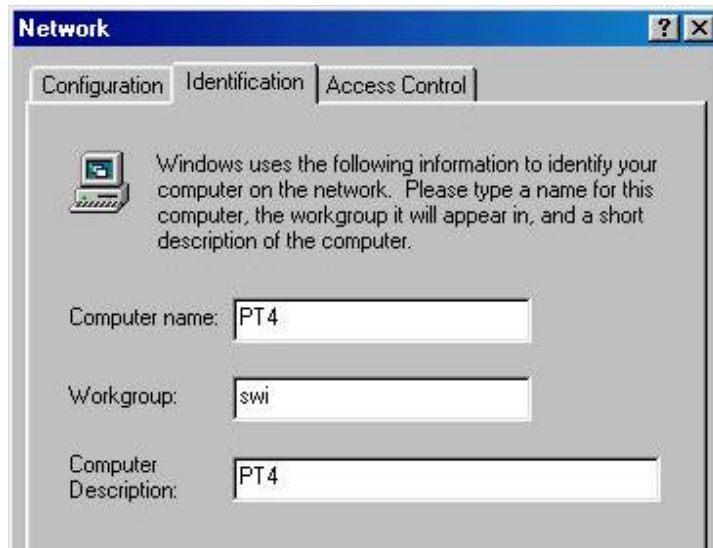


Figure 13.12.3.5 Identification

14. Finally, close the windows. Restart the ProtoTRAK.

15. On your computer, you want to make sure (using the same steps), that you have TCP/IP installed, matching workgroups, unique computer names, unique IP addresses that will work together (192.162.0.1 & 192.162.0.2 for example), Client for Microsoft Windows, and file and printer sharing enabled.
16. Restart both the computer and your ProtoTRAK. To verify that we have successfully networked both computers together, on your desktop double click on Network Neighborhood. You should see each other's computer name here. If not, contact SWI customer service or your systems administrator for technical support.

Now you will need to share and map at least one drive in order to be able to transfer files back and forth between two networked computers. Sharing a drive determines what folders on your computer you want to share with the rest of the network. Mapping sets up a drive on your computer as a shortcut to a shared folder across a network. Depending on your needs, you would most likely want to share a folder on your PC, and map it on your ProtoTRAK.

17. To share a drive / folder on your computer, first you must locate it using Windows Explorer. Double click on MY COMPUTER, then on C DRIVE. Find the folder you wish to share, then using the right mouse button click on it and select SHARING. You can also share the entire C drive if you'd like (making everything on your hard drive available for sharing). You should get a window like the one below.

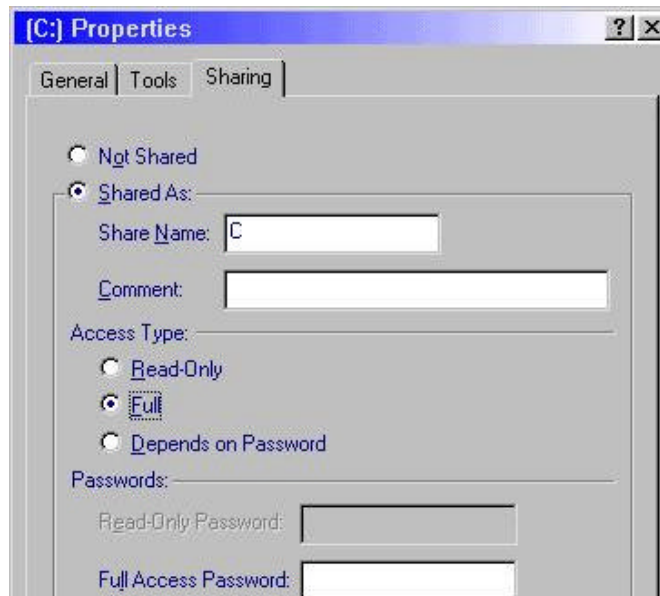


Figure 13.12.3.6 Sharing

18. Make sure you select the option "Shared As", and most likely you will want to select Full access type, so that you can open AND save files across the network. Click on OK when you are finished.
19. To map a drive on your ProtoTRAK, go into Windows 98, then double click on NETWORK NEIGHBORHOOD on your desktop. You should be able to see the name of the computer(s) that you are currently networked to. Double click on the one you wish to map to, and when opened you should see a list of all folders that we enabled

sharing. Right click on one of these folders, and select "Map Network Drive", you should get a window like the one below.



Figure 13.12.7 Mapping

20. Choose an available drive letter of your choice, make sure that the "Reconnect at logon" option is checked, and click OK. From now on, the drive letter that you chose will be assigned as a shortcut to the shared folder (which is actually a folder stored on the networked computer). On your ProtoTRAK, you will be able to see this drive when you go to PROG IN/OUT, and attempt to OPEN or SAVE your files. Using the TAB button to move to the LOOK IN window, you should now be able to save / retrieve to and from your newly mapped network drive.

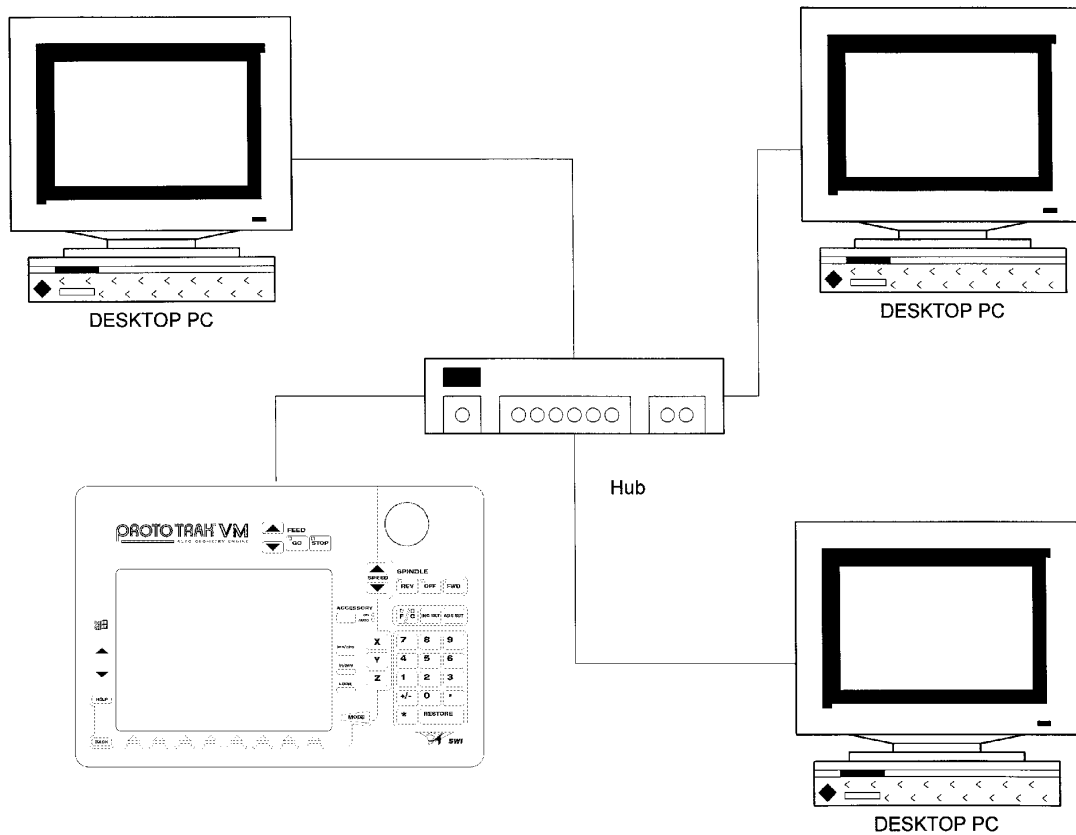


FIGURE 13.12.4 Peer-to-Peer (3 desktop PC's and 1 VM CNC control hooked to a hub)

13.12.4 Server and Client Network Overview

A more complex type of network, often found within an office environment, involves the use of a *server* and individual *clients*. Connecting more than two computers requires the addition of a hub.

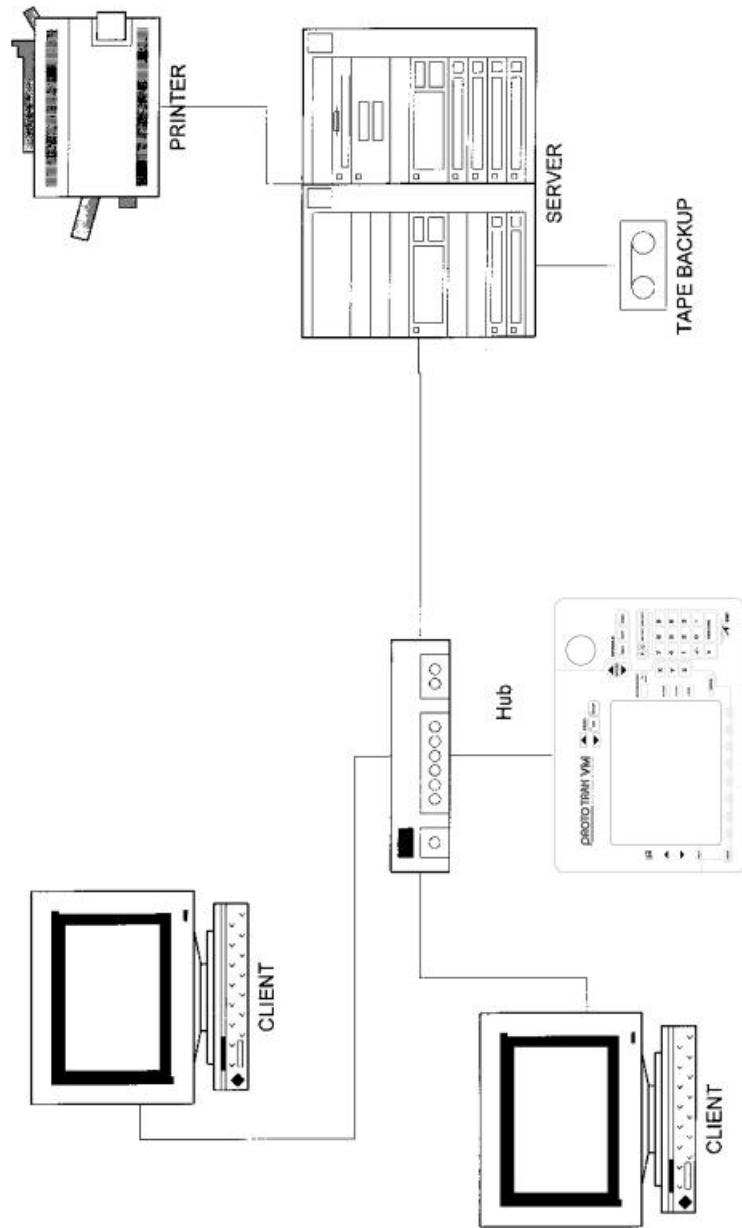
Server: a computer that is dedicated primarily to the task of providing shared resources, such as hard drives, and printers to other users on a network.

Client: a computer that has access to a network server and shares its resources.

Hub: a device that allows multiple computers to communicate with each other.

We recommend that you use a DHCP switch instead of a simple hub. The switch will automatically assign IP addresses, subnet masks, and gateway addresses in addition to performing the distribution functions of a hub.

This will make setting up multiple CNCs or computers easier. Simply use the default settings shown in Section 13.12.5 and plug the devices into the switch.



100847

FIGURE 13.12.5 Typical server and client set-up diagram

In many companies, a main database is often kept on the server and is accessed by the clients. The client can be thought of as an individual desktop computer in an office cubicle. When the person using a client needs to make a change to a main database, he accesses the server through the network.

The server uses software specifically designed to manage this type of network set-up. The most commonly used software is Microsoft Windows NT/2000. This software carries out the tasks of coordinating the access to shared network resources among the network client computers. Most clients use a desktop version of Microsoft Windows 95/98/ME/XP.

13.12.5 Network Description of the ProtoTRAK

The following table summarizes important settings for the ProtoTRAK V series and ProtoTRAK S series CNCs. It is intended for use by a professional Network Administrator.

Note: The ProtoTRAK has many similarities to a desktop computer, but it is somewhat different because the use of the computer's resources is optimized for running part programs and a machine in real time. In order to avoid causing a slow-down or even instability in the operating system of the control, keep the following in mind while setting up the network:

- Do not use a resource-intensive networking program such as SMS.
- Avoid loading in programs that direct background tasks, e.g. anti-virus programs.

System and Network Set Up	
Default password	PT4
Default user name	PT4
Default computer name	PT4SN0001
Default work group	SWI
Network settings	Client for MS Networks
	File and print sharing
Default protocols	Net beui
	TCP/IP
Network log in	Windows log in*
	Enabled share level access control
TCP/IP set up	Obtain IP address automatically
Dns	Disabled
Gateway	Not used
Wins configuration	Use DHCP for wins resolution

*do not change

13.13 RS232 Interface

While networking and floppy disks are more modern, reliable and convenient methods, the RS232 protocol may also be used to transfer files between the ProtoTRAK VM and other controls and computers. The ProtoTRAK VM will support RS232 file transfer for Open and Save operations in the Program In/Out Mode. The other operations (such as Back up) cannot be done through RS232.

13.13.1 Connections

To transfer a file between the ProtoTRAK VM and a remote computer or control, the two systems must be connected to each other with an RS232 cable. On the ProtoTRAK VM, plug the cable into the connector on the left side of the pendant (see Figure 3.2). On the remote computer or control, the cable may go to any available serial port. The baud rate for the ProtoTRAK VM is set at 4800; use Service Code 37 to reset the baud rate.

The pin-out for the 9-pin connector is the standard RS232 PC - PC Laplink/Interlink cable (PC 9 way female - PC 9 way female).

9D Female RS232 signal	Pin		Pin	9D Female RS232 signal
---------------------------	-----	--	-----	---------------------------

GND	Case	-----	Case	Gnd
TxD	3	-----	2	RxD
RxD	2	-----	3	TxD
RTS	7	-----	6	DSR
CTS	8	-----	4	DTR
DSR	6	-----	7	RTS
SG	5	-----	5	SG
DTR	4	-----	8	CTS

13.13.2 Receiving a File

Receiving a file via the RS232 is done through the file Open command. See Figure 13.12.1.

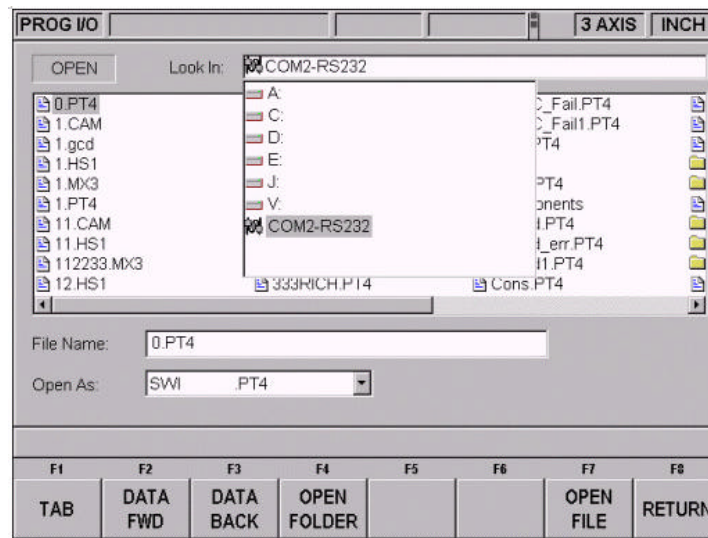


Figure 13.13.1 The RS232 port is selected in the Look In box.

Once the RS232 port is selected, pressing the Tab key will position the cursor in the now empty list of contents. A message window will appear prompting you to enter the file name of the file you wish to receive. Press Tab again to move the cursor to the File Name box. You may name the file the same name as the one it has on the remote computer or control, or you may give it a different name. After entering the file name, move the cursor to the Open As box and select the format of the file that you wish to receive as it exists in the remote computer or control. For example, if you were transferring a Haas file from a Haas CNC, you would select the file type: HS1.

After you have set up the ProtoTRAK VM to receive the file, press the Open soft key. A message window will appear to notify you that the ProtoTRAK VM is ready to receive the file.

If the remote computer or control is a TRAK or ProtoTRAK CNC (or is a computer that is running ProtoTRAK or TRAK A.G.E. offline software) you can send the file by simply using the Save softkey (via the RS232) in the Program In/Out Mode. If the remote computer or control is not a TRAK or ProtoTRAK CNC, consult your manual for sending files via the communications port.

Modern Windows operating systems have programs such as Windows hyper-terminal and CAD/CAM systems have the capability of communicating via the RS232 port.

The commands for an MS-DOS system are:

'mode COM1: 4800,e,7,1' – to set up the port where COM1 is the serial port on the remote computer
and 'copy "filename" COM1: ' – to send a copy of the file to the COM1 port.

As the program is received it will be converted into the ProtoTRAK VM format and put into the current memory. Once the transfer is complete, the Select a Mode screen will appear.

13.13.3 Sending a File

Sending a file via the RS232 is done through the file Save command. Transfer will take place only with a program that is in the ProtoTRAK VM's current memory.

As with receiving the file, the RS232 port must be selected in the Look In box.

Since the file name does not transfer with the program through RS232, you can ignore the file name box.

Place the cursor into the Save As box and select a file format. The ProtoTRAK VM will convert the file to the format that you select as it is being sent to the remote computer or control.

Once the transfer is set up, press the Save soft key. The ProtoTRAK VM will be placed in a ready state for transferring the file to the remote computer or control. A second Save key press is required to go from this ready state to actually sending.

The remote computer or control must be set up to receive the file. If the remote computer or control is a TRAK or ProtoTRAK CNC (or is a computer that is running ProtoTRAK or TRAK A.G.E. offline software) you can receive the file by simply using the Open softkey (via the RS232) in the Program In/Out Mode. If the remote computer or control is not a TRAK or ProtoTRAK CNC, consult your manual for sending files via the communications port.

Modern Windows operating systems have programs such as Windows hyper-terminal and CAD/CAM systems have the capability of communicating via the RS232 port.

The MS-DOS commands for receiving the file are:

'mode COM1: 4800,e,7,1'
and then: 'copy "filename" COM1:'

The receiving computer or control will attach the filename to the program it receives via the RS232 port.

When the receiving computer or control is ready, press the Save key a second time to send the file.

Once the file is sent, the Program In/Out Mode Save screen will remain and the Look In drive will change to the one used previous to the RS232 operation.

13.14 CAD/CAM and Post Processors

In addition to running G-code files, the ProtoTRAK will also accept CAM files and convert them into the ProtoTRAK events. This is a great advantage as it allows you to have your CAD/CAM programmer send files to the machine that the machinist can then work with in the familiar ProtoTRAK interface. The machinist can modify the program as necessary without having to go back to the CAD/CAM programmer.

In order to be able to convert the program from a CAM system to a ProtoTRAK program the CAM program must be two or 2½ axis. A 2½-axis program is defined as a program where the Z axis is stationary while X and Y is moving. If you want to run a full three-axis program, you should run a G Code, or .GCD program (see Section 13.11).

The above 2½-axis restriction does not mean that the ProtoTRAK is not capable of running three-axis simultaneous programs written in ProtoTRAK events (as some ill-informed competitors would have you believe). This restriction is a matter of practicality. Because the ProtoTRAK allows you to program in part geometry and therefore will figure out the tool path for you, the process of converting a three-axis program gives the ProtoTRAK a tool position problem that it cannot resolve without a lot more data from you. The other reason is that the output from a CAM systems for three-axis shapes is in the form of thousands and thousands of straight-line G01 moves that would convert into the equal number of ProtoTRAK Mill events. This is hardly a manageable program.

Instead of forcing the issue in a silly way, we give you the more elegant solution of running GCD files. To our competitors, we respectfully point out that the thread and helix milling canned cycles of the ProtoTRAK are obvious examples of three-axis simultaneous interpolation.

In order to run a CAM program, the program must be posted through a post-processor that makes some adjustments to the output of the CAM software so that it is understood by the ProtoTRAK. The ProtoTRAK uses a post-processor that is very similar to the Fanuc 6M.

If you are not familiar with writing a post-processor, we recommend that you contact your CAD/CAM supplier. We will be happy to work with him to get you the post-processor you need.

13.14.1 Writing a Post Processor

The following are modifications to a Fanuc 6 post-processor that are necessary for writing the ProtoTRAK post-processor.

Beginning file format: The ProtoTRAK has no special requirements, it does not need any special characters.

End of file format: The ProtoTRAK requires the % to show the end of the file. Characters after the % will be ignored.

Beginning of an operation: the ProtoTRAK requires that the tool number, feedrate and tool offset appear before, or on the same line, as a move command. In addition, the ProtoTRAK VM (but not the SM) requires the spindle speed be set. The absolute zero of the ProtoTRAK is set in a different mode and does not need to be set at the beginning of each operation. The feedrate is modal, once it is set, it remains the same until changed.

Lines: The line feed (or carriage return/line feed) signals the end of the line (ASCII code hex 0A or OD 0A). A semicolon is optional.

Coordinates: May be formatted in inch or metric. The addresses used for specifying coordinates are X, Y, Z, I, J, K. The valid ranges are:

- Inch: min -99.9999 to max +99.9999
- Mm: min -999.99 to max +99.999

Rapid moves: Rapid moves are generated by the ProtoTRAK automatically as part of the definition of an event. For this reason, G0 moves are discarded unless they specify a location other than the beginning of the following event.

Linear moves: G01 are formatted the same as rapid moves.

Arcs: Arc centers are specified by the address I, J and K for the X, Y and Z axes. The number following the I, J and K are incrementally referenced from the starting point of the arc. Radius values are not allowed.

Tool Numbers and Tool Changes: The format of the tool number is from T1 to T99. During program run, the ProtoTRAK will rapid to home for a tool change and pause for the tool to be loaded manually and the operator to press GO.

Feed rates: The ProtoTRAK is programmed in inches (or mm) per minute using the 'F' address.

Spindle speed: The ProtoTRAK SM does not have spindle speed control, so S values are ignored. The VM uses S values.

File name: Use the .CAM extension so the ProtoTRAK will recognize the file as a CAM file and convert it into ProtoTRAK events when it is opened. File names may include up to 20 alpha-numeric characters.

13.14.2 Convertible G-Codes

The following G-codes may be used in a CAM file that you want to have converted to a ProtoTRAK program. G Codes that are not on the list below have no correspondent operation in the ProtoTRAK events and will be ignored when the program is converted.

If a G Code is essential to your program and you do not see it here, you can do one of two things.

- Convert the file from CAM to ProtoTRAK and add an event to the resulting ProtoTRAK program.
- Run the program as a GCD file (See Section 13.11).

G Code	Description
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G20	Input in inch
G21	Input in metric
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G54	Work coordinate system 1 selection
G55	Work coordinate system 2 selection
G56	Work coordinate system 3 selection

G57	Work coordinate system 4 selection
G58	Work coordinate system 5 selection
G59	Work coordinate system 6 selection
G73	Peck drilling cycle
G80	Hole machining canned cycle cancel
G81	Drilling cycle, spot boring
G82	Drilling cycle, counter boring
G83	Face hole machining cycle
G84	Tapping canned cycle (VM only)
G85	Face boring cycle
G89	Boring cycle, dwell at bottom
G90	Absolute programming
G91	Incremental programming
G98	Return to initial point in canned cycle
G99	Return to point R in canned cycle

13.14.3 Supported Addresses

CAM information is communicated through the use of ADDRESS – WORD pairs. For example in the line "N01G0X1.Y2." N, G, X, and Y are addresses. The other information (01, 1, and 2) are Data Words. The line starts with the Address = N and the data word = 01. The N address is defined as meaning "LINE NUMBER", therefore N01 means Line # 1, and so on.

X, Y, Z	Dimensions along the specified axis
I, J, K	Distance to arc center I = X, J = Y, K = Z.
D	Tool Diameter
M	Miscellaneous Functions
G	Preparatory Function
H	Tool Length Offset Selector (silently ignored).
N	Line Number (silently ignored)
T	Tool Number
F	Feedrate
P	Dwell time for drill/bore canned cycles
L	Repetition count for drill/bore canned cycles
Q	Depth of cut for drill/bore canned cycles
R	Reference point for drill/bore canned cycles
S	Spindle Speed

13.14.4 Format Terms and Definitions

Number formats

- A. preparatory function number, denoted <prep-func>
 1. format: dd
 2. leading 0 suppression
 3. range: 0 to 99
- B. sequence or line number, denoted <seq-number>
 1. format (independent of units): dddd
 2. leading 0 suppression
 3. range: 1 to 9999
- C. Unsigned coordinate word, denoted <coord>
 1. format:

- metric: dddd.ddd
- inch: dddd.dddd
- 2. the "+" sign is implied and therefore may be omitted
- 3. leading 0 suppression
- 4. if no decimal point is given, the supplied number will be interpreted as an integer (i.e. a whole number).
- 5. Fractional portion is optional
- 6. Range:
 - metric: 0 to 99999.999
 - inch: 0 to 9999.9999
- D. signed coordinate word, denoted <scoord>
 - 1. format:
 - negative number: -<coord>
 - positive number: +<coord> or <coord>
 - 2. range:
 - metric: -99999.999 to 99999.999
 - inch: -9999.9999 to 9999.9999
- E. tool function, denoted <tool>
 - 1. format : dd (use 2-digit only)
 - 2. leading 0 suppression
 - 3. range: 1 to 99.
- F. miscellaneous or M codes function number, denoted <prep-func>
 - 1. format: dd
 - 2. leading 0 suppression
 - 3. range: 1 to 99
- G. feedrate values, denoted <frate>
 - 1. format:
 - metric: ddddd
 - inch: ddd.dd
 - 2. leading 0 suppression
 - 3. decimal point not required
 - 4. fractional portion is optional
 - 5. range:
 - metric: 1 to 6350
 - inch: 0.1 to 250
- H. RPM command (VM)
 - 1. format: dddd
 - S1000 = 1000 RPM

13.4.5 G Codes that Generate Errors

G Code	Function
G27	Reference point return check
G28	Return to reference point
G29	Return from reference point
G30	Return to 2 nd reference point
G31	Skip function
G33	Thread cutting
G37	Tool length automatic measurement
G38	Cutter radius compensation vector change
G39	Cutter radius compensation corner rounding
G45	Tool offset increase

G46	Tool offset decrease
G47	Tool offset double increase
G48	Tool offset double decrease
G62	Automatic corner override mode
G63	Tapping mode
G65	User macro simple call
G66	User macro modal call
G67	User macro modal call cancel
G74	Counter tapping cycle
G76	Fine boring
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G92	Programming of absolute zero point
G95	Feed per revolution

13.14.6 Accepted M Codes

M Code	Function
M00	A pause is generated. The axes will not move, but the motors will be engaged. The spindle motor will not turn off.
M02	Executed automatically at the end of all programs. Turns off the servo motors and all auxiliary functions. The auxiliary function box must be present for this function to work.
M05	Stops the spindle at the end of the current event. The auxiliary function box must be present for this function to work.
M06	Tool change. The M06 is ignored, as the tool change on the ProtoTRAK is accomplished by changing the tool number.
M07	Flood coolant on. This will turn on the auxiliary box A/C outlet before the event.
M08	Mist coolant on. This will turn on the air supply from the auxiliary box before the event.
M09	Coolant off. This will turn off the auxiliary box A/C outlet and air supply after the event.
M12 & M20	Send a pause to the indexer and wait for an "in position" response.
All other M codes will be ignored.	
<i>Notes:</i>	
<ol style="list-style-type: none"> 1. Place M Codes on same line as movement G Code. 2. One M Code per block. 	

Sample Program No. 1

Basic 2 ½ - Axis Program

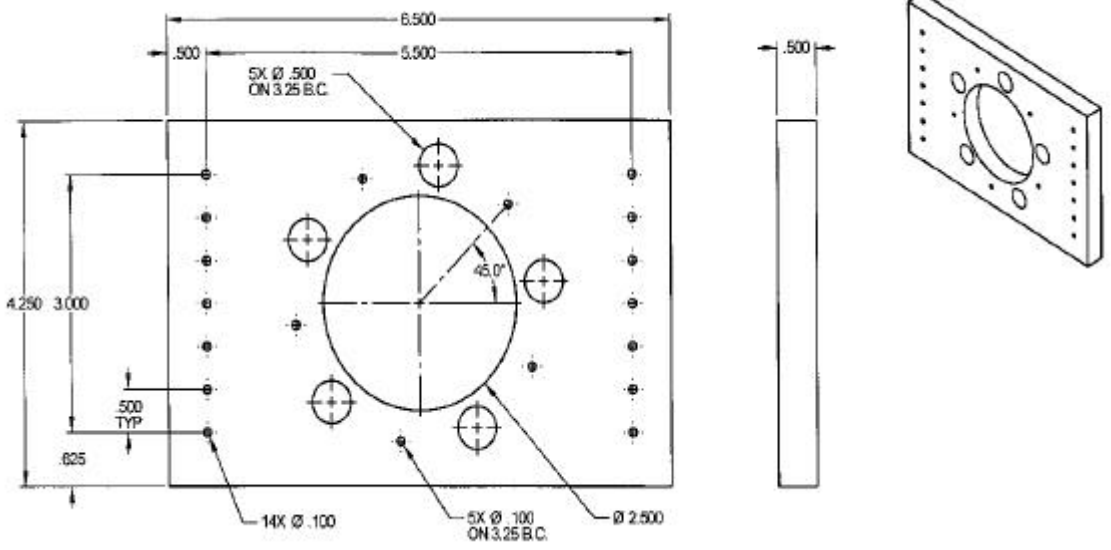
This program is designed to give you practice on programming 2½-axis events.

The Program

All programs start by first selecting "Program" from the front panel softkeys. You may enter in an alphanumeric program description, or simply press "Go to Begin" to get started.

Start your program by selecting Pocket and then Circle Pocket.

Sample Drawing 1



EVENT 1	CIRC PCKT	NOTES
X Center	abs/set	Sets the pocket center to X zero
Y Center	abs/set	Sets the pocket center to Y zero
Z Rapid	.1 abs	Sets the Rapid plane to 0.1"
Z End	-.6 abs	Sets the pocket depth to -0.6"
Radius	1.25 abs	Sets radius of pocket to 1.25"
Direction	2 abs	Makes the cut direction CCW
# of Passes	2 abs	Cuts the pocket using two (2) depths
Entry Mode	1 abs	Makes the tool ramp into the material
Fin Cut	.01 abs	Takes a 0.01" cut to finish the pocket
RPM	2500 abs	Sets the pocket cutting rpm to 2500 rpm
Fin RPM	3000 abs	Sets the pocket finish rpm to 3000 rpm
Z Feedrate	5 abs	Sets the ramp feedrate to 5 ipm
XYZ Feedrate	20 abs	Sets the pocket cutting feedrate to 20 ipm
Fin Feedrate	15 abs	Sets the finish pocket feedrate to 15 ipm.
Tool #	abs/set	Sets tool to default # 1

EVENT 2	Bolt Hole	NOTES – drill for 0.1" bolt pattern
Drill or Bore	1	Selects Drill Function
# of Holes	5 abs	Known print value
X Center	abs	Defaults to X zero
Y Center	abs	Defaults to Y zero
Z Rapid	.1 abs	Sets the rapid plane to 0.1"
Z End	-.6 abs	Sets the drill depth to -0.6"
Radius	1.625 abs	The radius of the bolt hole circle
Angle	45 abs	Angle of first hole from zero (0) degrees
# of Pecks for Drill	6 abs	Sets drill function for 6 pecks
RPM	2000 abs	Sets tool rpm to 2000
Z Feedrate	10 abs	Sets Z plunge rate to 10 ipm
Tool #	2 abs	Selects Tool # 2 (0.1" drill)

EVENT 3	DRILL	NOTES – center drill for 0.1" hole patterns
Drill or Bore	1 abs	Selects Drill Function
X	2.75 abs	Known print value
Y	-1.5 abs	Known print value
Z Rapid	.1 abs	Sets the rapid plane to 0.1"
Z End	-.6 abs	Sets the drill depth to -0.1"
# of Pecks for Drill	6 abs	Sets drill function for 6 pecks
RPM	3000 abs	Sets tool rpm to 3000
Z Feedrate	5 abs	Sets Z plunge rate to 5 ipm
Tool #	inc/set	Keeps Tool # 2 (0.1" drill)

Select More then Copy, then Repeat (to do the drilling using the previous event info)

EVENT 4	COPY REPEAT	NOTES – for the drill 0.1" operation
First Event #	3 abs	Selects the first event to copy from
Last Event #	3 abs	Selects the last event to copy to
X Offset	inc/set	Does not offset the pattern in X
Y Offset	.5 inc/set	Offset the pattern in Y by 0.5"
Z Rapid Offset	inc/set	Does not change the Z Rapid plane
Z Offset	inc/set	Does not change the Z depth
# Repeats	6 abs	Copies the selected events 6 times
% RPM	inc/set	Leaves the rpm where it was
% Feed	inc/set	Leaves the feedrate where it was
Tool #	inc/set	Leaves the previous tool as the current tool

Select More then Copy, then Repeat (to do the drilling using Event # 3 thru 9 info)

EVENT 10	COPY REPEAT	NOTES – for the drill 0.1" operation
First Event #	3 abs	Selects the first event to copy from
Last Event #	9 abs	Selects the last event to copy to
X Offset	-5.5 inc/set	Offset the pattern in X by -5.5"
Y Offset	inc/set	Does not offset the pattern in Y
Z Rapid Offset	inc/set	Does not change the Z Rapid plane
Z Offset	inc/set	Does not change the Z depth
# Repeats	1 inc/set	Copies the selected events 1 time
% RPM	inc/set	Leaves the rpm where it was
% Feed	inc/set	Leaves the feedrate where it was
Tool #	inc/set	Leaves the previous tool as the current tool

EVENT 17	BOLT HOTE	NOTES – drill for 0.5" bolt pattern
Drill or Bore	1	Selects Drill Function
# of Holes	5 abs	Known print value
X Center	abs	Defaults to X zero
Y Center	abs	Defaults to Y zero
Z Rapid	.1 abs	Sets the rapid plane to 0.1"
Z End	-.75 abs	Sets the drill depth to -0.75"
Radius	1.625 abs	The radius of the bolt hole circle
Angle	9 abs	Angle of first hole from zero (0) degrees
# of Pecks for Drill	6 abs	Sets drill function for 6 pecks
RPM	1500 abs	Sets tool rpm to 1500
Z Feedrate	10 abs	Sets Z plunge rate to 10 ipm
Tool #	3 abs	Selects Tool # 3 (0.5" drill)

This is the end of the program.

Sample Program No. 2

2-Axis Profile with Limited Print Data

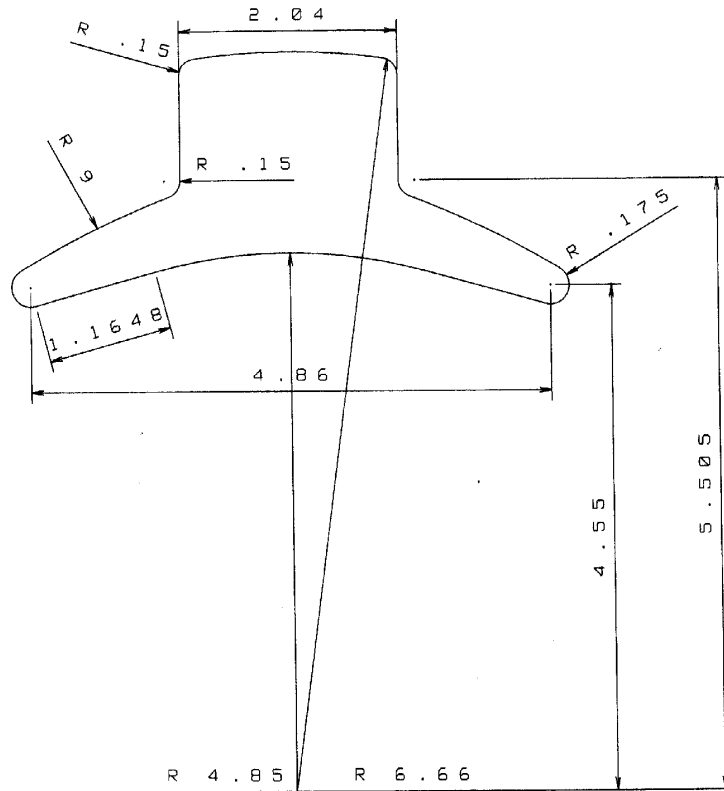
This program is designed to provide practice with the ProtoTRAK VM A.G.E. programming system. A basic rule of thumb for A.G.E. programming:

- When you have unknown tangent elements, you may skip data.
- When you have unknown non-tangent elements, you use the Guess feature.

The Program All programs start by first selecting "Program" from the front panel softkeys. You may enter in an alphanumeric program description, or simply press "Go to Begin" to get started.

Be certain to start your program by selecting Profile.

Sample Drawing 2



i00851

EVENT 1	IRREGULAR PROFILE	NOTES
X Begin Y Begin Tool Offset Finish Cut RPM Fin RPM Feedrate Fin Feedrate Tool #	-1. abs 6.66 abs 2 Inc Set 1500 Inc Set 100 abs Inc Set abs/set	Leads into part – no witness mark Leads into part – no witness mark Sets tool offset LEFT Means no finish cut Sets RPM No Finish RPM Sets feedrate to 100 ipm. Means no finish feedrate Sets tool to default # 1

EVENT 2	AGE Mill	NOTES
Tangent X End Y End Conrad Angle End Length Line Angle	1 0 6.66 abs	Selects Tan Mill Known print value Known print value Press Page Forward now, as no further data is required

EVENT 3	AGE ARC	NOTES
Tangent Direction X End Y End X Center Y Center Conrad Radius Chord Length Chord Angle	1 1 abs 1.02 abs Guess (6.6) abs Abs/set Abs/set .15	Selects Tan Arc Selects CW arc direction Known endpoint Unknown endpoint, guess Defaults to 0 – known print dimension Defaults to 0 – known print dimension Sets Conrad of 0.15" Press Page Forward now, as no further data is required

EVENT 4	AGE MILL	NOTES
Tangent X End Y End Conrad Angle End Length Line Angle	1 Inc set 5.505 abs	Selects tan mill By default, X will remain the same Known print dimension Press Page Forward now, as no further data is required

EVENTS	AGE ARC	NOTES
Tangent Direction X End Y End X Center Y Center Conrad Radius Chord Length Chord Angle	1 2 data forward data forward 1.17 abs 5.505 abs data forward 0.15 abs	Selects tan arc CCW arc direction Unknown Endpoint Unknown Endpoint Known Center Known Center Known print dimension Press Page Forward now

EVENT 6	AGE ARC	NOTES
Tangent	1	Selects tan arc
Direction	1	CW arc
X End	Data Forward	Unknown data
Y End	Data Forward	Unknown Data
X Center	Data Forward	Unknown Data
Y Center	Data Forward	Unknown data
Conrad	Data Forward	
Radius	9 abs	Known print dimension
Chord Length		Press Page Forward now, as no further data is required
Chord Angle		

EVENT 7	AGE ARC	NOTES
Tangent	1	Selects tan arc
Direction	1	CW
X End	data forward	Unknown value – not required
Y End	data forward	Unknown value – not required
X Center	2.43 abs	Known print dimension
Y Center	4.55 abs	Known print dimension
Conrad	data forward	
Radius	.175 abs	Known print dimension
Chord Length		Press Page Forward now, as no further data is required
Chord Angle		

EVENT 8	AGE MILL	NOTES
Tangent	1	Selects tan mill
X End		Press Page Forward now, as no further data is required
Y End		
Conrad		
Angle End		
Length		
Line Angle		

Event 9	AGE ARC	NOTES
Tangent	1	Selects tan arc
Direction	2	CCW
X End	data forward	Unknown value – not required
Y End	data forward	Unknown value – not required
X Center	0 abs	Known print dimension
Y Center	0 abs	Known print dimension
Conrad	data forward	
Radius	4.85 abs	Known print dimension
Chord Length		Press Page Forward now, as no further data is required
Chord Angle		

EVENT 10	AGE MILL	NOTES
Tangent	1	Selects tan mill
X End		Press Page Forward now, as no further data is required
Y End		
Conrad		
Angle End		
Length		
Line Angle		

EVENT 11	AGE ARC	NOTES
Tangent Direction X End Y End X Center Y Center Conrad Radius Chord Length Chord Angle	1 1 data forward data forward -2.43 abs 4.55 abs data forward 0.175 abs	Selects tan arc CW Unknown value – not required Unknown value – not required Known print dimension Known print dimension Known print dimension Press Page Forward now, as no further data is required

EVENT 12	AGE ARC	NOTES
Tangent Direction X End Y End X Center Y Center Conrad Radius Chord Length Chord Angle	1 1 Data Forward Data Forward Data Forward Data Forward Data Forward 9 abs	Selects tan arc CW arc Unknown data Unknown Data Unknown Data Unknown data Radius of Arc Press Page Forward now, as no further data is required

EVENT 13	AGE ARC	NOTES
Tangent Direction X End Y End X Center Y Center Conrad Radius Chord Length Chord Angle	1 2 -1.02 abs 5.505 abs -1.17 abs 5.505 abs	Selects tan arc CCW Known print dimension Known print dimension Known print dimension Known print dimension Press Page Forward now, as no further data is required

EVENT 14	AGE MILL	NOTES
Tangent X End Y End Conrad Angle End Length Line Angle	1 0 inc Guess 6.6 abs 0.15	Selects tan mill X value is unchanged Unknown value – Guess Known print value Press Page Forward now, as no further data is required

EVENT 15	AGE ARC	NOTES
Tangent	1	Selects tan arc
Direction	1	CW
X End	0 abs	Known print dimension
Y End	6.66 abs	Known print dimension
X Center	0 abs	Known print dimension
Y Center	0 abs	Known print dimension
Conrad		Press Page Forward now, as no further data is required
Radius		
Chord Length		
Chord Angle		

EVENT 16	AGE MILL	NOTES
Tangent	1	Selects tan mill (off the part)
X End	1 abs	Known value
Y End	0 inc	Value unchanged
Conrad		Press Page Forward now, as no further data is required
Angle End		
Length		
Line Angle		

Press the Look Key Now! If all is OK, press Look key again to return to program, and End the AGE.

Checklist

In order for SWI to process a trainee for certification, all aspects of the training on the product must be successfully completed to the satisfaction of the trainer and the trainee. Please fill out and sign this document when the training is completed and send it to Southwestern Industries. Upon receipt, the certification document will be created and sent to the trainee.

Complete the following information:

Full name of person to receive certification _____

Company name and complete address _____

Name of trainer _____

Machine Name _____

Machine Model _____

Machine Serial No. _____

Place a checkmark next to each of the completed training areas:

<input type="checkbox"/>	1.	Definitions, terms and concepts for Axes, Planes and Referenced Data
<input type="checkbox"/>	2.	The DRO Mode, and using the machine tool manually
<input type="checkbox"/>	3.	Programming; using all canned cycles
<input type="checkbox"/>	4.	Editing capabilities
<input type="checkbox"/>	5.	Using all of the program storage and retrieval choices
<input type="checkbox"/>	6.	Converting data from other program sources
<input type="checkbox"/>	7.	Setting up tools and using the tool table
<input type="checkbox"/>	8.	Programming, setting up and running a complete part
<input type="checkbox"/>	9.	Understanding networking capabilities
<input type="checkbox"/>	10.	Using Math Help
<input type="checkbox"/>	11.	Using AGE capability in programming profiles
<input type="checkbox"/>	12.	Using Windows functions for non programming functions
<input type="checkbox"/>	13.	A review of related SWI products and compatibility to the VM control

 Trainee Signature Date

 Trainer Signature Date

ProtoTRAK VM Training

Southwestern Industries, Inc

TRAK

Warranty Policy

Warranty

TRAK products are warranted to the original purchaser to be free from defects in workmanship and materials for the following periods:

Product	Warranty Period	
	Materials	Factory Labor
New TRAK	1 Year	1 Year
Any EXCHANGE Unit	90 Days	90 Days

The warranty period starts on the date of the invoice to the original purchaser from Southwestern Industries, Inc. (SWI) or their authorized distributor.

If a unit under warranty fails, it will be repaired or exchanged at our option for a properly functioning unit in similar or better condition. Such repairs or exchanges will be made FOB Factory/Los Angeles or the location of our nearest factory representative or authorized distributor.

Disclaimers of Warranties

- This warranty is expressly in lieu of any other warranties, express or implied, including any implied warranty of merchantability or fitness for a particular purpose, and of any other obligations or liability on the part of SWI (or any producing entity, if different).
- Warranty repairs/exchanges do not cover incidental costs such as installation, labor, freight, etc.
- SWI is not responsible for consequential damages from use or misuse of any of its products.
- TRAK products are precision mechanical/electromechanical measurement systems and must be given the reasonable care that these types of instruments require:
- Replacement of chip scrapers and wipers is the responsibility of the customer. Consequently, the warranty does not apply if chips have been allowed to enter the mechanism.
- Accidental damage, beyond the control of SWI, is not covered by the warranty. Thus, the warranty does not apply if an instrument has been abused, dropped, hit, disassembled or opened.
- Improper installation by or at the direction of the customer in such a way that the product consequently fails, is considered to be beyond the control of the manufacturer and outside the scope of the warranty.