TRAK® Machining Centers
ProtoTRAK® RMX CNC
Safety, Programming, Operating and Care Manual

Document:	31550
Version:	121522

Covers Current Models:
- **TRAK VMC2**
- **Toolroom Machining Centers**
  - ○ **TRAK TMC5**
  - ○ **TRAK TMC7**
  - ○ **TRAK TMC10**
  - ○ **TRAK TMC12**
  - ○ **TRAK TMC14**

Covers Previously Branded Models:
- **TRAK VMC5**
- **TRAK VMC7**
- **TRAK VMC10**

TRAK MACHINE TOOLS

SOUTHWESTERN INDUSTRIES, INC.

2615 Homestead Place
Rancho Dominguez, CA 90220-5610 USA
T | 310.608.4422 | F | 310.764.2668
Service Department: 800.367.3165
e-mail: sales@trakmt.com | service@trakmt.com | web: www.trakmt.com

TRAK Machine Tools
Southwestern Industries, Inc.
2615 Homestead Place
Rancho Dominguez, CA 90220-5610
Phone 310.608.4422 ▪ Fax 310.764.2668
Service Department
Phone 800.367.3165 ▪ Fax 310.886.8029
Web: www.trakmt.com

Table of Contents

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

1.0 Introduction

Congratulations! Your TRAK Machining Center with the ProtoTRAK RMX CNC is a brand-new versatile machine that works in two ways: Toolroom Ops – when you need it to work like a TRAK Bed Mill, but with chips and coolant contained and Production Ops – when you need to run small to medium production quantities.

The amazing new ProtoTRAK RMX CNC delivers powerful capability with an easy-to-use interface, the most advanced control ever made for toolroom and small production milling. The ProtoTRAK RMX represents our decades of deep involvement in the world of making parts in small quantities. It transforms the best of technology into the ultimate tool for the people who make things on the shop floor.

The **Touch Screen** Interface enables you to interact with your programs and set ups with more certainty and control than ever before.

Defaults allow you to customize the ProtoTRAK RMX for how *you* make parts; they are easy to set and easy to change. See Section 6.5.

The **large LCD screen** and **fly out windows** allow you work with everything you need at the same time without flipping between screens.

Enhanced ProtoTRAK Assistance (EPA) is always available for you to quickly look up the information you need for what you are doing at the time. See Section 6.3.
With a little practice, you will find that the ProtoTRAK RMX is the easiest to use, most powerful ProtoTRAK ever…and that is saying a lot!

Manual Machining is always available and made easier with features like power feed, rapid positioning, 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.

1.1 Using This Manual

This manual provides enough information for most users in most situations. If you ever need more information or clarification, we want to help.
- Call us at 800-421-6875 and ask for Applications
- Visit our website http://www.trakmt.com
- Talk to your local ProtoTRAK representative

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 VMC2, TRAK TMC's, and the ProtoTRAK RMX CNC. Machine Control Options are described in this section.

Section 4 describes the operation of the toolroom machining center and some basic operations of the ProtoTRAK RMX CNC.

Section 5 defines some terms and concepts useful in learning to program and operate the ProtoTRAK RMX CNC.

Section 6 describes the innovative Information Keys including Defaults, Options and EPA (Enhanced ProtoTRAK Assistance).

Section 7 describes the powerful capabilities in DRO mode that will make your manual work more productive than ever.

Section 8 Programming, Part 1: Covers some general programming information and Instructions on starting new programs.

Section 9 Programming, Part 2: Program Events - instructions for the canned cycles, or events, used to program the ProtoTRAK RMX CNC.

Section 10 Programming, Part 3: the A.G.E., or Auto Geometry Engine, so powerful it gets its own section.

Section 11 describes the capabilities in Edit Mode such as copying from clipboard and G-Code editor.

Section 12 describes Set-up Mode and has information for the user about Verify Part, viewing Tool Path for the program created, and other routines to get the program ready to run.

Section 13 Run Mode provides instruction on running the program in TRAKing or CNC mode.

Section 14 describes how to use Program In/Out Mode to save programs created or open existing programs, and lists all of the file extensions currently supported.

Section 15 describes the 4th Axis: 4th Axis events, how it works in DRO mode, G-Codes, and how to setup the 4th Axis.

Note - Any mention of "TMC" or "TMC's" throughout the manual refers to the following TMC models: TMC5, TMC7, TMC10, TMC12, TMC14.

2.0   Safety

The safe operation of the TRAK VMC2/TMC depends on its proper use and the precautions taken by each operator.

- Read and study this manual and the "ProtoTRAK VMC2 Safety, Installation, Maintenance, Service and Parts List Manual" or/and "ProtoTRAK TMC Safety, Installation, Maintenance, Service and Parts List Manual." Be certain every operator understands the operation and safety requirements of this machine before its use.

- Never run the machine with enclosure doors open when in production ops.

- 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 Machining Centers and Automatic, Numerically Controlled Milling, Drilling and Boring Machines (ANSI B11.23-2002) (R2007). Available from The American National Standards Institute, 1819 L Street N.W., Washington D.C. 20036

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.

Safety & Information Labels Used on TRAK VMC2 Machines

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

⚠ **SAFETY INSTRUCTIONS** ⚠

1. Read and understand the Operator's Manual and all warnings on this sign before operating machine.

2. Machine should only be operated by qualified personnel that have been trained in the operation and use of this machine.

3. Machine starts and moves automatically. Never place any part of your body near or on moving parts of this machine.

4. Stop spindle completely before touching the tool, work piece, or spindle.

5. Do not operate machine unless all guards, interlocks, and safety devices are installed and functioning.

6. Always clamp work piece and tool securely. Avoid excessive feeds and spindle speeds.

7. Wear safety glasses, shoes, and hearing protection when operating machine.

8. Remove rings, watches, jewelry and loose fitting clothes. Keep hair away from moving parts of the machine.

9. Installation and service must be performed by qualified personnel only, following instructions in the Maintenance Manual. Turn off and lock-out the power at the main electrical panel before servicing.

10. It is the responsibility of the user to be sure that this machine is in safe operating condition at all times, and that the operator follows the safe operating procedures described in the Operator and Maintenance Manuals.

Do not remove or disfigure this sign.

S-S1140

TRAK MACHINE TOOLS
SOUTHWESTERN INDUSTRIES, INC.
Service: 800-367-3165
Office: 800-421-6875
www.TRAKMT.com

⚠ **SAFETY WARNING!** ⚠

1. Hazardous voltage present in the electrical equipment of this machine. Only qualified engineers are allowed to install, test, adjust or maintain it.

2. Isolate power before install, test, adjust or maintain the electrical equipment of this machine.

3. Do Not turn on the power before the protective grounding has been securely connected.

4. Do Not change any device of this machine without permission.

FAILURE TO COMPLY WITH THE ABOVE MAY RESULT IN A SERIOUS ACCIDENT

S-S1875

⚠ **WARNING**

1. Never open door during operation.
2. Do not operate with door removed.

Failure to observe the above may result in serious injury.

S-S3120

⚠ **WARNING**

Keep hands, clothing and body clear of rotating spindle. Machine starts and moves automatically. Can cause severe injury.

1. Set control in manual mode before working near spindle.
2. Isolate machine before servicing.

S-S3010

Safety & Information Labels Used on TRAK VMC2 Machines

WARNING: PRESSURIZED VESSEL

THIS AIR TANK STAYS PRESSURIZED EVEN IF THE MAIN AIR IS DISCONNECTED. BEFORE DISCONNECTING THIS TANK, FIRST DISCONNECT THE MAIN AIR SUPPLY. LOWER THE FRONT DOOR ALL THE WAY DOWN, THEN RELEASE ALL AIR PRESSURE FROM THE TANK BY SLOWLY OPENING THE DRAIN VALVE LOCATED ON THE BOTTOM OF THE TANK.

CAUTION!
IN ADDITION TO THE COMPRESSED AIR, THIS TANK MAY CONTAIN SOME OIL AND WATER. USE PROPER EYE PROTECTION AND EXERCISE CAUTION WHEN OPENING THE DRAIN VALVE.

WARNING

1. IMPROPER OPERATION OR IMPROPER MAINTENANCE OF THIS MACHINE MAY CAUSE SERIOUS BODILY INJURY OR DEATH.

2. DO NOT OPERATE THIS MACHINE UNTIL YOU HAVE RECEIVED OPERATING AND SAFETY INSTRUCTIONS FROM YOUR EMPLOYER.

3. FOLLOW ALL SAFETY PRACTICES PRESCRIBED BY YOUR EMPLOYER AND OUTLINED IN THE "SAFETY, PROGRAMMING AND OPERATING MANUAL".

4. IT IS THE RESPONSIBILITY OF THE EMPLOYER TO PROVIDE AND ENSURE METHODS OF SAFEGUARDING PER ANSI B11.22-2002 (R 2007) & B11.23-2002 (R 2007)

5. THE MANUFACTURER IS NOT LIABLE (RESPONSIBLE) FOR ANY DAMAGE OR INJURY OF ANY KIND TO PERSONS OR PROPERTY CAUSED BY OR RESULTING FROM THE IMPROPER OR UNAUTHORIZED USE, OPERATION, MAINTENANCE, ALTERATION, MODIFICATION, CHANGE IN CONFIGURATION OF THIS MACHINE OR ANY OF ITS COMPONENTS, PARTS, OR THE USE OF THIS UNIT WITH ANY THIRD PARTY ACCESSORIES OR PARTS.

DO NOT REMOVE THIS SIGN FROM MACHINE S-S 1170

VISE & FIXTURE WARNING

Vise or fixture must not protrude beyond the back edge of the worktable to avoid collision with the ATC. It may not protrude more than 1.5 inches beyond the front of the worktable to avoid collision with the front door. Use SWI modified Kurt vise P/N VISE-2OPM10 or Kurt vise 3600V-INT or equivalent

MAX
1.5 MAX
FRONT
S-S1250

DOOR OPEN **DOOR CLOSE**
S-S1260

FASTER SLOWER FASTER SLOWER

1 2 5 6
3 4 7 8
S-S1150

5
TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

Safety & Information Labels Used on TRAK VMC2 Machines

LIFT HERE

⚠ **CAUTION**

Check and fill drawbar intensifier monthly
Recommended oil:
ISO 32

⚠ **CAUTION**

1. Air pressure must be between 6 - 7kgf/cm² (85psi-100psi).

2. Filter and Lubricator must be maintained once a week.

3. Recommended Air line – 12mm (1/2 inch).

S-S2050

NOTICE

Carefully hold tooling before pressing the tool release button. Otherwise, the spindle, tooling, table and workpiece could be damaged.

S-S3062

ISO VG 32
Slideway Oil
ONLY

S-S1121

208 VOLTS

WARNING - HOT!

Eye protection must be worn

MAINTENANCE

ONCE A MONTH, PLEASE REMOVE THIS BACK COVER AND VACUUM OUT THE MAJORITY OF THE CHIPS IN THE ATC COMPARTMENT AREA TO PREVENT EXCESSIVE CHIP BUILDUP, WHICH MAY OTHERWISE RESULT IN A POTENTIAL ATC MALFUNCTION OR FAULT.

S-S1220

Safety & Information Labels Used on TRAK VMC2 Machines

MAINTENANCE SCHEDULE

DAILY

- Remove majority of chips from around the axis slide ways, work table and way covers, especially between table and ATC.
- Empty chip tray.
- Visually check lubrication pump oil level and make sure it is always above the minimum line.
- Visually check the coolant level and add if it is low.
- Visually check the air regulator filter.

MONTHLY OR AS SPECIFIED

- Remove rear ATC and Y-axis motor compartment access covers and clean chips and other debris. Cleanup any oil or coolant accumulated on the bottom panel of the machine base.
- Visually inspect the condition of way and ball screw covers. Clean if showing chip build up.
- Check and if needed replace the air regulator filter element.
- Every two months drain and remove the coolant tank and clean inside, including pump screen. Fill with new coolant.
- Visually check the tool unclamp oil reservoir level.
- Remove all air filters in the electrical cabinet and transformer enclosure every two months and clean.
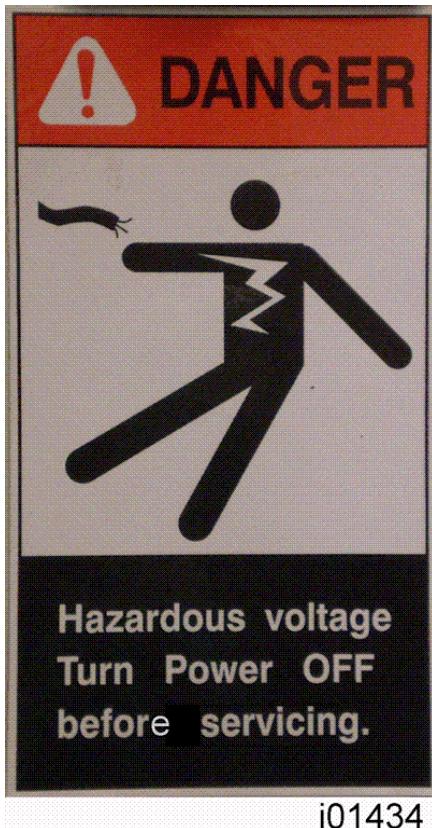
YEARLY

- Remove all covers. Vacuum chips, debris, and wipe down machine from top to bottom.
- Check backlash on each axis and adjust if needed. Refer to the machine manual for more information.
- Inspect machine for any unusual wear and play.
- Check cables and pneumatic lines for any excessive abrasions or cuts.
- Inspect the ATC air cylinder and grease the ATC linear rail bearing blocks.
- Please contact SWI Service Dept. for help with any machine maintenance procedures. Service Dept. contact number is 1-800-367-3165.

i27694-20

Safety & Information Labels Used on TRAK TMC Machines

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

DANGER

Hazardous voltage
Turn Power OFF
before servicing.

i01434

i01441

DANGER

Entanglement
Hazard.
Keep your hands
away from the
running tooling..

i01437

Safety & Information Labels Used on TRAK TMC Machines

i01438

i01435

i01439

Safety & Information Labels Used on TRAK TMC Machines

Air pressure setting

Max.pressure setting	115psi / $8kg/cm^2$
Min.pressure setting	70psi / $5kg/cm^2$
Pressure sensor setting	60psi / 4 kg/cm²

i01436

SAFETY INSTRUCTIONS

1. Read and understand Operator's Manual and all warnings on the sign before operating. Failure to follow these instructions and warnings can result in serious injury or death.
2. This machine starts and moves automatically. Never place any part of your body near or on moving parts of this machine.
3. Always stop the spindle compeletly before touching the work piece, tool or spindle.
4. Do not operate this machine unless all guards, interlocks and other safety devices are in place and functioning.
5. Always clamp work piece and cutting tool securely. avoid excessive feeds and spindle speeds.
6. Remove rings, watches, jewelry and loose fitting clothing. Keep your hair away from moving parts of the machine.
7. Always wear safety glasses, safety shoes and hearing protection when operating this machine.
8. Service or installation of this machine must be performed by qualifed personnel only, following procedures described in the Maintenance Manual. Turn off and lock out power at main electrical panel before servicing.

It is the responsibility of the user to be sure that this machine is in safe operating condition at all times and that the operator follows the safe operating procedures described in the Operator and Maintenance Manuals and all signs attached to this machine. If you have any questions concerning the safe operation of this machine, contact your supervisor or local Distributor.

Please do not remove or disfigure this sign.

i01440

Safety & Information Labels Used on TRAK TMC Machines

WARNING

- IMPROPER OPERATION OR IMPROPER MAINTENANCE OF THIS MACHINE MAY CAUSE SERIOUS BODILY INJURY.
- DO NOT OPERATE THIS MACHINE UNTIL YOU HAVE RECEIVED OPERATING AND SAFETY INSTRUCTIONS FROM YOUR EMPLOYER.
- FOLLOW ALL SAFETY PRACTICES PRESCRIBED BY YOUR EMPLOYER AND OUTLINED IN THE PROGRAMMING, OPERATING AND CARE MANUAL.
- ALWAYS AVOID UNSAFE OPERATING CONDITIONS SUCH AS EXCESSIVE FEEDS AND SPEEDS.
- ALWAYS BE SURE ALL SAFETY DEVICES ARE IN GOOD OPERATING CONDITION (SEE MANUAL).
- ALWAYS BE CERTAIN THAT SERVICE IS PERFORMED ONLY BY QUALIFIED PERSONNEL.
- IT IS THE RESPONSIBILITY OF THE EMPLOYER TO PROVIDE AND ENSURE POINT OF SAFEGUARDING PER ANSI B11.6-1984 (R1994) & ANSI B11.8-1983 (R1994).
- THE MANUFACTURER IS NOT LIABLE (RESPONSIBLE) FOR ANY DAMAGE OR INJURY OF ANY KIND TO PERSONS OR PROPERTY CAUSED BY OR RESULTING FROM THE IMPROPER OR UNAUTHORIZED USE, OPERATION, MAINTENANCE, ALTERATION, MODIFICATION, CHANGE IN CONFIGURATION OF THIS MACHINE OR ANY OF ITS COMPONENTS, PARTS, OR THE USE OF THIS UNIT WITH ANY THIRD PARTY ACCESSORIES OR PARTS.

DO NOT REMOVE THIS SIGN FROM MACHINE.

i22774

DANGER

Pinch point watch your hands.

Safety & Information Labels Used on TRAK Machines

TRAK Serial Number Plate will have the specific information engraved applicable to the mill.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

2.3 Safety Precautions

1. Do not operate this machine before the TRAK VMC2 and TMC Installation, Maintenance, Service & Parts List Manuals, and ProtoTRAK VMC2/TRAK TMC 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, 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:
    o Before changing tools.
    o Before changing parts.
    o Before you clear away the chips, oil or coolant. Always use a chip scraper or brush.
    o Before you make an adjustment to the part, fixture, coolant nozzle or take measurements.
    o 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 compressed air to remove the chips or clean the machine.

12. Disconnect power to the machine before you change belts, pulley, or gears.

13. Keep work areas 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, tool changer, 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. After an emergency stop, release the E-stop and press the power reset button for 2 seconds to turn the servos back on.

21. 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.

22. Check the direction (+ or -) of movement of the table, saddle, and quill when using the jog or power feed or ram out.

23. 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 tool for the job.

24. Large overhang on cutting tools when not required result in accidents and damaged parts.

25. Inspect the retention knobs for damage or excessive wear before each use.

26. 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. Keep flammable materials and fluids away from the machine and hot, flying chips.

27. Always be certain that you know if the machine is in production or toolroom Operation.

28. Always be certain the door is closed during Run Mode when the machine is in production or toolroom Operation.

29. When working in manual mode (not CNC) make sure the computer control is switched to DRO or OFF.

30. Always slide z-axis hand wheel to the retracted position when not in use.
    Note - *This safety precaution is not applicable to the VMC2.*

31. Make sure coolant lines and nozzles are facing away from the operator.

32. Move the X and Y hand wheels to prevent hitting while loading tooling, parts and tools to prevent accidental movement of axis.
    Note - *This safety precaution is not applicable to the VMC2*.

33. Use only the retention knobs that each machine is designed to use. The use of the incorrect retention knobs or incorrect usage of the retention knobs may result in injury or property damage.

3.0   Description

3.1   ProtoTRAK RMX Control Specifications

The ProtoTRAK RMX CNC provides you with an extraordinary combination of power and ease of use. Programming and operations are kept simple and at any moment you may select Help Screens or make changes to the programming defaults using the Info Keys.

3.1.1   List of System Specifications

Pendant Control Hardware

- 2 or 3-axis CNC, 3-axis DRO
- Front panel electronic handwheel for manual operation
- 15.6" Touchscreen LCD
- Intel® 2.0 GHz processor
- 4 GB Ram
- At least 64 GB of mSATA SSD
- 4 USB connectors
- 2 Ethernet Ports (1 for user and 1 for motion control)
- Override of program Feedrate
- Override of spindle speed
- LED status lights built into run panel
- E-stop
- Toggle switch for Production Ops and Tool Room Ops
- Spindle Control (FWD, REV, OFF)
- Feed STOP and GO
- Fine vs Course resolution control for Tool Room EHWs (**Note -** *Not available on the VMC2.*)
- X, Y, Z, and 4th axis selection for Front Panel EHW, as well as 4 levels of resolution control
- Separate buttons to toggle COOLANT, AIR, and Coolant Through Spindle in ON, OFF, or AUTO modes (**Note -** *Coolant Through Spindle feature not available on the VMC2.*)
- Buttons for WASH and DOOR LOCK on & off. (**Note -** *Not available on the VMC2.*)
- Chip Removal auger FWD and REV buttons available. (**Note -** *Not available on the VMC2.*)
- POWER RESET button to energize the servos
- 3.5mm Headphone jack for video sound (headphone supplied)

Computer Module Control Hardware

- 4 Axis motor control – X, Y, Z, and 4th axis
- 26 inputs
- 16 outputs
- Mill Indexer Interface

Software Features – General Operation

- Enhanced ProtoTRAK Assistance (EPA)
  - Context responsive help with control operations
  - Step-by-step instructions, diagrams and videos
- "Watch Me" videos for quick introduction to ProtoTRAK RMX operation.
- "What's New" videos for quick review of upgraded features in new releases
- Clear, uncluttered screen display
- Fly out windows for quick access to features and info
- Programming Defaults to simplify part programming
- Individual User profiles for Defaults
- Event Options to modify Defaults or select additional functionality

15

- QWERTY touchscreen keyboard
- Calculator
- Prompted data inputs
- English language – no codes
- Soft keys – change within context
- Windows® operating system
- Selectable two or three-axis CNC
- Color graphics with adjustable view
- Gestures for pan, zoom, rotate
- Inch/mm conversion
- Convenient modes of operation
- Networking

Info Soft Keys
- Status shows current state of the ProtoTRAK RMX
- Tool Table for instant access to tool set ups
- EPA (Enhanced ProtoTRAK Assistance) help information for ProtoTRAK RMX operation
- Math Help to easily calculate missing print data
- Options appear when available while programming
- Defaults to customize the programming style
- Key board to enter alphanumeric and special symbols
- Calculator for simple calculations
- Warnings appear when you must resolve a condition

Status Features
- IN/MM selection
- 2/3-axis selection
- Check current program name
- Check User Profile Name
- Check software and firmware versions
- Purchased Options Status: Advanced Features w/ Verify, DXF Converter, Parasolid Converter, Cam Out Converter, Auxiliary Functions, Clear Off Event, MTConnect, Electronic Handwheels, and 4th Axis. (**Note -** *Electronic Handwheels and 4th Axis are not available on the VMC2.*)
- Selectable Features (Turn On/Off Control Options): Advanced Features w/ Verify, Networking, MTConnect, Electronic Handwheels, and 4th Axis (**Note -** *Electronic Handwheels and 4th Axis are not available on the VMC2.*)

Tool Table Features
- Tool Library for using tools in multiple programs and DRO
- Program Specific Library to manage tools in the current program
- ATC Library to manage tools residing in the ATC
- All buttons related to ATC such as CALL TOOL and RETURN TOOL available within Tool Table
- Convenient tool length reference to Base tool
- Tool type reference
- Tool material reference
- # Flutes reference
- Tool Diameter set
- Diameter modifier set
- Z modifier set
- Ram Jog and manual quill enabled for quick tool height reference

EPA (Enhanced ProtoTRAK Assistance) Features
- Context sensitive topics for ProtoTRAK RMX control operations
- Keyword search
- Brief explanations of specific operations
- Screen shots for clarity
- Videos for more advanced explanations
- Updated with software release versions

Math Help Features
- 22 separate intuitive routines
- Calculate missing print data using data provided
- Screen Graphics to guide data inputs
- Feed – IPM / MMPM conversions
- Speed – RPM / SFPM conversions
- Cartesian / Polar conversions
- Right triangle calculations
- Load Beg – loads solution as X and Y Begin
- Load End – Loads solution as X and Y End
- Load Center – Loads solution as X and Y Center

Programming Defaults
- Instant access to all defaults via fly out window
- Feedrate per Minute or per Tooth
- Spindle speed RPM or Surface Speed
- Peck type for drill events
- Number of Passes for Profile, Pocket, Island and Clear Off events
- Finish Cut for Profile, Pocket, Island and Clear Off events
- Step over percentages
- Face mill tool path
- Z entry: plunge, helical or zig zag (ramp)
- Adaptive Tool Path selections
- Numerous default parameters for pocket, Island and Clear Off events
- Speed and feed override for subroutine and copy events
- Forward or Backward Mirror Direction
- X and Y Home Positions
- Inch or MM
- 2 or 3 axis CNC at start up
- Maximum rapid feedrates – Tool Room & Production Ops
- Cut Tolerances
- Tool Compensation
- Parasolid Set Z Offsets
- Z Rapid
- Part Change Location
- Individual Default User Profiles
- Engrave Event – XY Reference
- Subroutine and Copy Events – Z entry method, Tool Path Pattern, Finish Cut
- Z Retract (***Note –*** *Default available on the VMC2 only.*)

Keyboard Features
- QWERTY style

- Touchscreen operation
- Numbers
- Special characters: $ @ # ? % ( ) < = * + -
- Caps Lock

DRO Mode Features
- Incremental and Absolute dimensions
- Graphical Override of Axis Feed and Spindle RPM
- Jog at rapid with override
- Powerfeed X, Y or Z
- Teach-in of manual moves
- Programmable Go To dimensions
- Servo return to 0 absolute
- Tool offsets from Tool Library
- Line Center calculation
- Circle Center calculation
- Tool #

Program Mode Features
- Circular Interpolation
- Linear interpolation
- Advanced Adaptive Tool Path
- Geometry programming
- Toolpath programming
- Auto Geometry Engine – Built-in CAD to fill in missing print data for you while you program
- Alphanumeric program names
- Automatic Scaling of print data
- Nesting
- Multiple Fixtures
- Incremental and absolute dimensions may even be mixed on a single point
- Automatic diameter cutter comp
- Program data editing
- Part graphics update while programming
- Selectable display between size of drawing and number of events
- List Step graphics relate Events to Drawing
- Editing of programmed data
- Swipe to move through programmed events

Options Within Applicable Events
- Z Safety Plane
- Spindle Speed RPM or Surface Speed
- Feed rate per minute or tooth
- Event Comments on/off
- # of Passes or Depth per Pass
- Bottom Finish Cut
- Cutting method One Way or Zig-Zag
- Step-over %
- Machining Angle in XY (0 – 90 deg.)
- Drill Peck: Variable, Fixed, Chip Break
- Dwell Request

- Multiple Holes
- Z Entry Plunge, Zig-Zag or Helical
- Tool path Pattern: Offset, Parallel
- Tool path Pattern: Adaptive
- Order of Passes: Depth or Region
- Cut Tolerances
- Rest Milling Options

Canned Cycles (Event Types)
- Position
- Drill / Bore / Tap / Helical Drill
- Bolt Hole Drill / Bore / Tap / Helical Drill
- Mill
- Arc
- Circular profile
- Rectangular profile
- Irregular Profile (with Auto Geometry Engine)
- Face Mill
- Circle pocket
- Rectangular pocket
- Irregular Pocket (with Auto Geometry Engine)
- Islands including Pocket and Island shapes
- Subroutine Repeat / Mirror / Rotate
- Copy – Mirror / Rotate / Convert Drill to Tap
- Helix
- Engrave – Horizontal / Radial / Vertical / Mirror
- Thread Mill
- Program Pause
- Clear Off (O)

Edit Mode Features
- Group Delete of Events
- Search Edit to make changes to multiple events
- Erase current program
- G-Code Editor
- Clipboard to copy events for insertion elsewhere
- Update GCD when g code program is loaded

Set-Up Mode Features
- Toolpath graphics with selectable views
- Estimated Run Time clock
- Step forward and backward through of Toolpath graphics
- XYZ tool position locations displayed in Step through mode
- Verify Make Part – step by step tool path verification
- Verify View Part – solid model of finished part
- Fixture offsets – convenient table for managing fixtures with DRO and Jog
- Service Codes for infrequent machine and control settings
- Pictures – Add pictures to be saved with your ProtoTRAK or G Code files
- Notes – Add user notes to be saved with your ProtoTRAK or G Code files

Run Mode Features

- 3D CAM file run
- 3D G code file run
- Override of Programmed Axis Feed and Spindle Speed from 0 – 150%
- Virtual handwheels to control Override
- Tap override of Axis Feed and Spindle RPM
- Real time conversion of programmed RPM / SFM
- Real time conversion of programmed IPM / IPT
- Status display:
- Event #
- Fixture#
- Control System Ready
- Current Tool #
- Repeat #
- Countdown clock to next pause or tool change
- Event / G Code Comments
- Start at beginning
- Start at any Event
- Start at tool # for GCD programs
- Start at rough or finish pass
- Start at any finish cut for pockets and islands – XY, Z, XY & Z
- Show Absolute Dimensions during run
- Show Incremental Dimensions
- Show Tool Path – real time graphics with Tool icon
- Show Program
- TRAKing® – you control programmed X, Y and Z feeds with Electronic Handwheel
- Chip Clear

Program In/Out Mode Features

- Program storage to USB device plugged into Control Pendant
- Program storage to Network via RJ45 Port
- Program storage to optional SSD drive (O)
- Browse file locations
- Create New Folders
- CAM program converter
- Convert prior-generation ProtoTRAK programs to current (.PT10)
- Save Temp – saves all current programs, tools and other settings
- Open Temp – opens the data saved at last "Save Temp"
- Rename programs
- Cut, Copy, Delete, Paste of program(s)
- Look – Graphics preview without opening
- .PTG – saves GCD files with pictures & notes
- .MPF – saves as Siemens Sinumerik One G-Code file

Auxiliary Functions Option

Enables programming and control of:
- Coolant
- Air
- Pulse Indexer (O)
- Part Change Position
- Programmable Auxiliary Output to control secondary external device (O)
- Coolant through Spindle (CTS)

DXF File Converter Option
- Import/convert CAD data into ProtoTRAK programs
- DXF or DWG files
- Chaining
- Add or remove geometry
- Automatic Gap Closing
- Layer control
- Easy, prompted process you can do right at the machine
- Island and Profile Events

Parasolid File Converter Option
- Import and convert 2.5D CAD data into ProtoTRAK programs
- .X_T file support
- X, Y & Z dimensions are transferred into program events
- 2D and 3D views of part
- Chaining
- Add or remove geometry
- Easy, prompted process you can do right at the machine
- Island, Pocket, and Profile Events

Front Panel Electronic Handwheel
- Standard on all TMC's
- TRAKing® of programs during program run
- Can be used with Tool Room EHW
- Go to Dimensions in DRO Mode

Offline Programming Option
- ProtoTRAK RMX user interface for Windows PC
- Program parts and simulate CNC Run
- Modify files from current and former ProtoTRAK models

3.2　ProtoTRAK RMX Options

You may purchase software options with the original control purchase or later. If you purchased them with your original order, they should come pre-installed from our factory. You can easily tell if you have an option activated on your control by pressing the Status Key. You can also see all the available options by referring to Service Code 318 (see below).

To purchase Software Options, call our Sales Support Team at 800-421-6875 for pricing and to place your order. Before you install the option, we recommend that you first assure that you are running the most recent version of the ProtoTRAK RMX.

Once you purchase the option, the Sales Support Rep will give you an Activation Code.
1. Go to the Set-up Mode.
2. Select Serv Codes.
3. Enter 318, Inc Set or Abs Set.
4. Tap the option you want to activate from the list.
5. When prompted, enter the Activation Code.

3.2.1　Electronic Handwheels

Electronic Handwheels can be used for TRAKing®, manual Go To, and Fine / Course handwheel resolution. It replaces the direct manual control of the ball screw with an encoder that sends a signal to the computer, which then commands the appropriate response from the axis motor. Electronic Handwheels are active only in DRO Mode, Set up Mode to set tools and reference locations and Run Mode for TRAKing® and Chip Clear. Otherwise, the machine will not respond to the movement of the handwheels.

You may easily turn the Electronic Handwheels on or off through the Status Information Screen under Selectable Features.

Note - *This feature is not available on the VMC2.*

3.2.2　DXF File Converter Option

The DXF File Converter Option gives you powerful capability for quickly and easily translating DXF and DWG files into ProtoTRAK RMX programs. If you work with CAD drawings, we highly recommend the DXF file converter.

The DXF File Converter Option has its own manual which is shipped with the software. You may also view a copy of the manual on our web site at www.trakmt.com.

3.2.3　Parasolid File Converter Option

The Parasolid File Converter Option gives you powerful capability for quickly and easily translating Parasolid files (X_T) into ProtoTRAK RMX programs. If you work with 3D Solid Model CAD drawings, we highly recommend the Parasolid file converter.

The Parasolid Option has its own manual which is shipped with the software. You may also view a copy of the manual on our web site at www.trakmt.com.

3.2.4 CAM Out File Converter Option

The CAM out Option gives you the ability to create or modify a program using the easy, intuitive ProtoTRAK interface and then convert that program into a CAM file or G Code file to use to run other CNCs. Select PROG IN/OUT mode to save your program as a .CAM extension.

Note – *You do not need the CAM Out option to share programs between ProtoTRAK CNCs.*

3.2.5 Auxiliary Function Option

The Auxiliary Function Option allows you to have programmable auxiliary output to control a secondary external device. It also allows you to have a programmable pulse indexer which functions by sending out a command from the ProtoTRAK to the indexer to take over and then waits for a signal from the indexer to continue with ProtoTRAK machining.

3.2.6 MTConnect Function Option

The ProtoTRAK currently supports the MTConnect protocol which is designed for the exchange of data between shop floor equipment and software applications used for monitoring and data analysis. It is referred to as a read-only standard, meaning that it only defines the extraction (reading) of data from control devices, not the writing of data to a control device. Using MTConnect by itself will not provide you with a visual representation of the data gathered, therefore, a 3rd party software (eg: TrakHound) would be required.

In order for MTConnect to operate with ProtoTRAK, it requires the latest ProtoTRAK software version 2.2.0+ and internet connection via an Ethernet cable or USB wi-fi adapter.

When enabled, TRAK's MTConnect Option will allow users to monitor their machine's status, receive notifications, and review their machine's history to provide the following benefits:

- Enable machines to provide more efficient operations, improved production optimization, and increased productivity.
- Provide domain-specific vocabulary and data models eliminating the confusion of handling multiple proprietary formats
- Easily integrate with other standards by design.

3.2.7 Clear Off Option

Clear Off option gives you ability to program Clear Off Events which is an easy way to be able to clear off certain materials of your part by defining boundaries.

3.2.8 Siemens Sinumerik One (OUT) Option

The Siemens Sinumerik One (OUT) Option allows you to create or modify a program using the easy, intuitive ProtoTRAK interface and then convert that program into an MPF file to run on VMCsi machines. Select PROG IN/OUT mode to save your program as a .MPF extension.

3.3   Display Pendant
3.3.1   Programming Panel
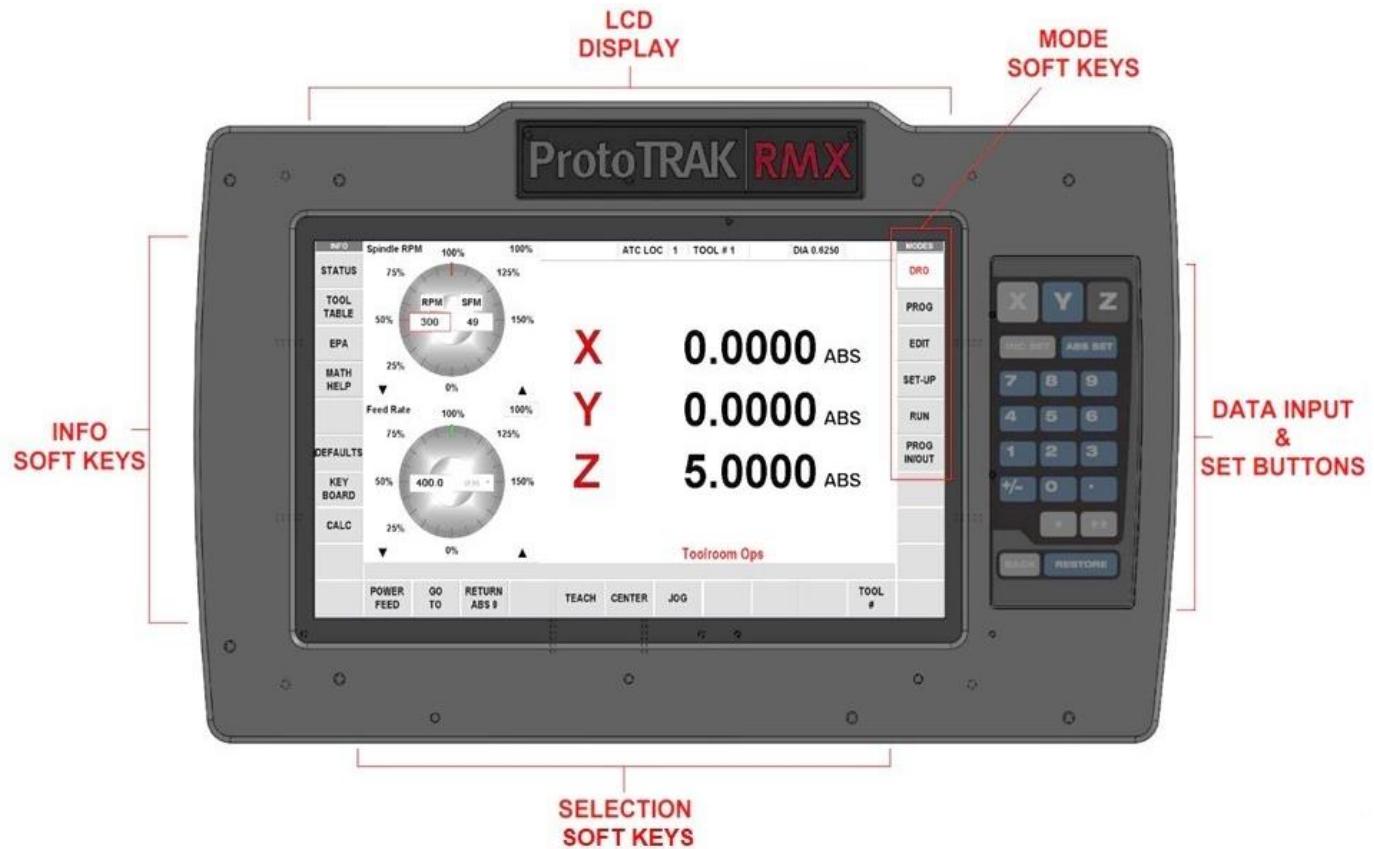
Figure 3.3.1 - Programming Panel – The upper panel on the front of the pendant

Keyboard Hard Keys

X, Y, Z: Selects axis for subsequent commands

INC SET: Loads incremental dimensions and general data

ABS SET: Loads absolute dimensions and general data

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

RESTORE: Clears an entry, aborts a keying procedure

BACK: Moves back within the screen or the DIL when there isn't a better way to do it.

* KEY: Used to capture screenshot at any given time to help troubleshoot

** KEY: Not used at this time.

3.3.2   Run Panel
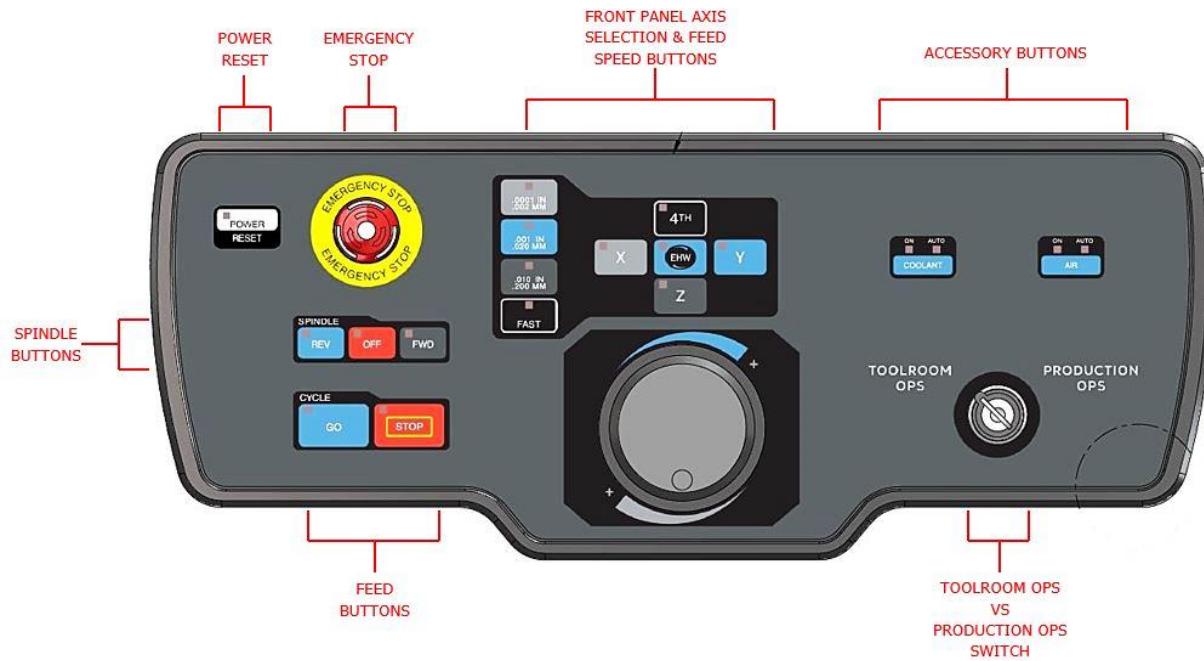
Figure 3.3.2a - VMC2 Run Panel – The lower panel on the front of the pendant.
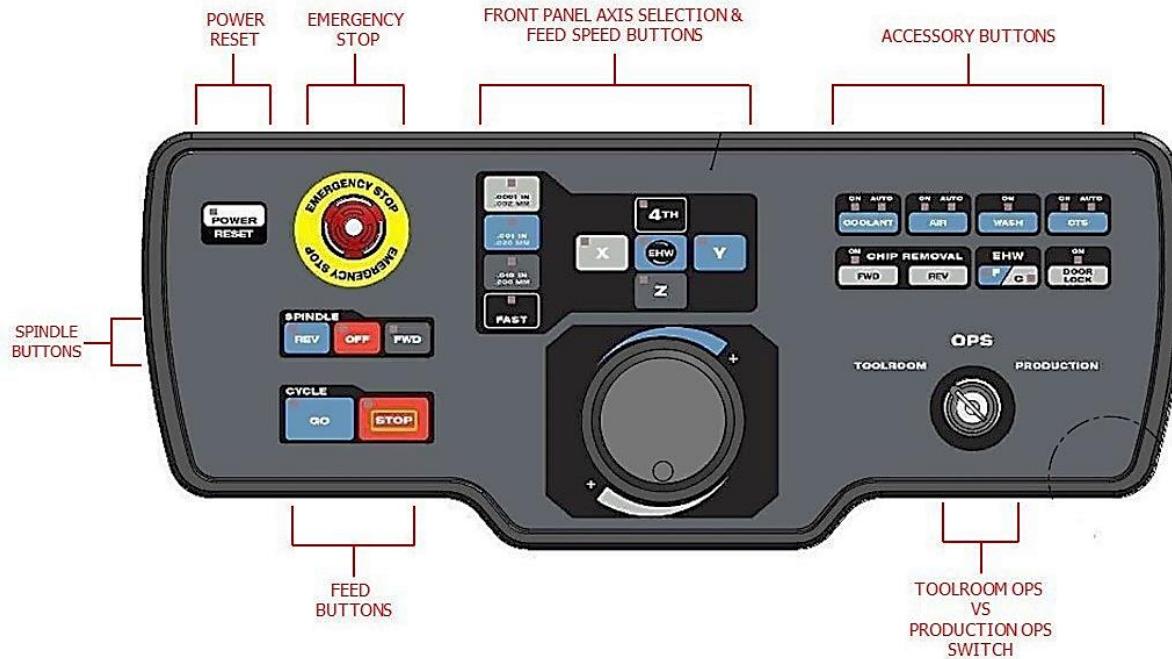
Figure 3.3.2b - TMC Run Panel – The lower panel on the front of the pendant.

POWER / RESET: The LED on indicates there is power to the servo motors. After starting the ProtoTRAK, Press and hold this key until the white LED comes on to initiate. This button will need to be pressed any time the control gets into a critical fault condition after start up and after an emergency stop.

EMERGENCY STOP: The emergency stop (E-stop) switch kills all power to the spindle and ProtoTRAK's

servomotors. The computer and pendant remain powered. To resume working from an emergency stop, you must twist to release the emergency stop button and press the power reset button.

You may also want to press in the Emergency Stop at the end of the day if you would like to leave the machine on but prevent the operation of the coolant pump or the Electronic Handwheels.

You must reset the power after each Emergency stop by pressing and holding the Power/Reset hard key.

Spindle Keys:
REV: Runs the spindle in reverse at programmed speed with any override.
OFF: Turns spindle off and servo brakes to a stop.
FWD: Runs spindle in the forward direction at programmed spindle speed with any override.

Feed Keys:
GO: Initiates motion in Run. The green LED on the GO key will be lit when the servomotors are moving the machine 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.

Front Panel Electronic Handwheel Keys:
Front Panel Electronic Handwheel: Used to control X, Y, Z axis positioning, spindle rpm override, and axis feed override. There are 100 positions or detents per revolution.
FP EHW: Enables handwheel to control positioning along X, Y, Z axes.
X, Y, Z: Selects axis to be controlled by the Electronic Handwheel.
.0001in / .002mm: Selects incremental axis motion per handwheel detent. One revolution equals (.0001in x 100) 0.01in.
.001in / .02mm: Selects incremental axis motion per handwheel detent. One revolution equals (.001in x 100) 0.1in.
.01in / .2mm: Selects incremental axis motion per handwheel detent. One revolution equals (.01in x 100) 1.0in.
FAST: Selects fast axis motion for coarse movement up to 300 ipm.

Accessory Buttons:
COOLANT: Press to turn coolant on (ON LED is lit). When it's in the ON position, the flood coolant pump will come on and stay on during machining operations. It will turn off during tool changes. Press and hold the key to turn coolant on auto (AUTO LED is lit). In Auto mode, the coolant pump will be controlled as programmed via the aux event. Press the key again to turn coolant off. If neither light is on, the coolant pump will not operate. The coolant will turn off at tool changes automatically. The user does not need to program this functionality.

AIR: Controls air from spindle to help clear chips. Press to turn air on (ON LED is lit). When it's in the ON position, the air will come on and stay on during machining operations. It will turn off during tool changes. Press and hold the key to turn air on auto (AUTO LED is lit). In Auto mode, the air will be controlled as programmed by the aux event. Press the key again to turn air off. If neither light is on, the air will not operate. The air will turn off at tool changes automatically.

WASH: Press key to activate chip wash to help move chips down to the chip auger location. (***Note -*** *Not available on the VMC2.*)
CTS: Press to turn on Coolant through Spindle (ON LED is lit). When it's in the ON position, the coolant through spindle will come on and stay on during machining operations. It will turn off during tool

changes. Press and hold the key to turn coolant on auto (AUTO LED is lit). In Auto mode, the coolant through spindle will be controlled as programmed via the aux event. Press the key again to turn CTS off. If neither light is on, the coolant through spindle will not operate. The CTS will turn off at tool changes and also when door is open automatically. The user does not need to program this functionality. (**Note -** *Not available on the VMC2.*)

CHIP REMOVAL: Press FWD key to activate chip auger, press again to turn off. Press and hold REV key to drive chip auger in reverse to help clear jams (make sure FWD is off). (**Note -** *Not available on the VMC2.*)

EHW FINE/COURSE:
Selects the resolution for the Optional Electronic Handwheels. (**Note -** *Not available on the VMC2.*)
C: Moves the table at 0.800" per revolution for X,
moves the saddle at 0.800" per revolution for Y.

F: Moves the table at 0.200" per revolution for X,
moves the saddle at 0.200" per revolution for Y.

DOOR LOCK: Activates the door lock mechanism whenever a program or other automatic move is running. The LED light on the key is on when the door lock feature is active (it is possible for the door to be unlocked with the light being on). (**Note -** *Not available on the VMC2.*)

TOOL ROOM OPS vs PRODUCTION OPS SWITCH: Turn key to the left to run the machine in toolroom ops. Turn key to the right to run in production ops. While in DRO and RUN mode, the OPS selected will be displayed in red font on the lower right-hand section of the screen.
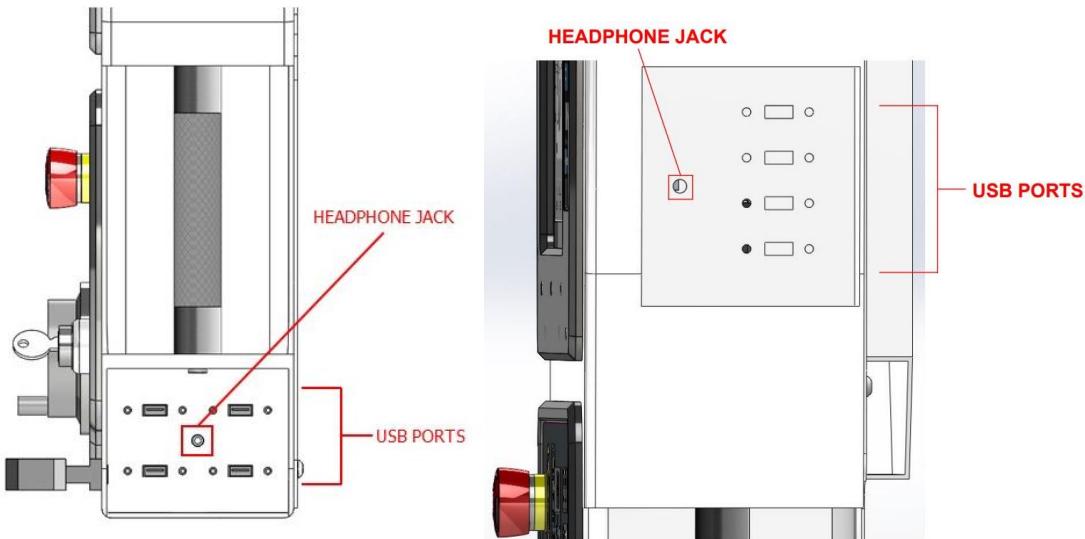
Keyboard Soft Keys
Once the ProtoTRAK RMX pendant has booted up, the soft keys are enabled and operated by the touchscreen.

Fly Out Windows
Touching one of the INFO Soft Keys initiates another window to open (or "fly out"). When one of these windows is active, the associated INFO Soft Key will be yellow. In order to put the window away, or switch to another Fly Out Window, press the INFO Soft Key again.

Warnings are also presented in a Fly Out Window. When a Warning appears, you must press the Clear soft key to dismiss the warning.

3.3.3   Pendant Right Side

Right side of the VMC2 Pendant   *Right side of the TMC Pendant*

Inset on the right side of the pendant are four USB ports for the thumb drive, keyboard, mouse, etc. and a headphone jack.

Installing and Using the USB Thumb Drive Flash Memory
The first time you install the USB Thumb Drive, we recommend that you install it after the ProtoTRAK RMX has booted up. Once it is installed, the memory will be accessible on Drive D or E Drive if you have the internal memory option. We highly recommend that you use the high-quality Delkin® brand thumb drive we provide with the USB Memory Option.
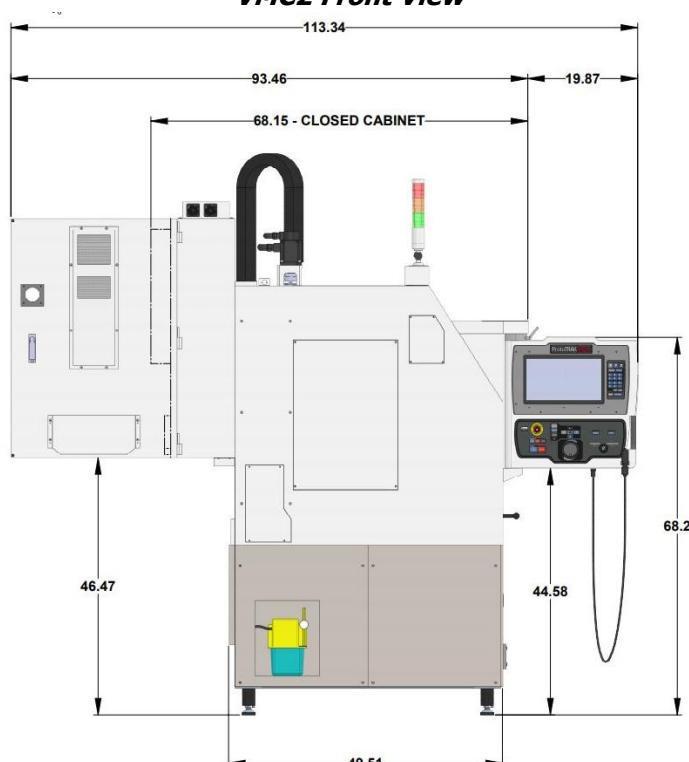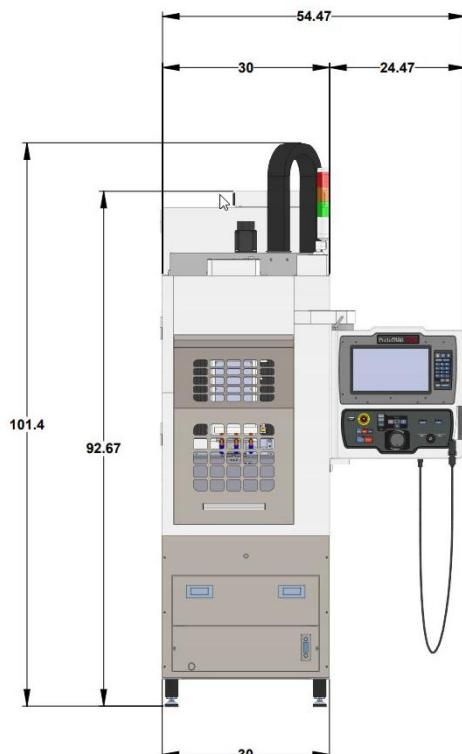
Otherwise, thumb drives are readily available in a variety of stores. While not as reliable, customers also have success with the SanDisk® brand. Other brands may require the installation of separate drivers, a hassle you could probably do without.
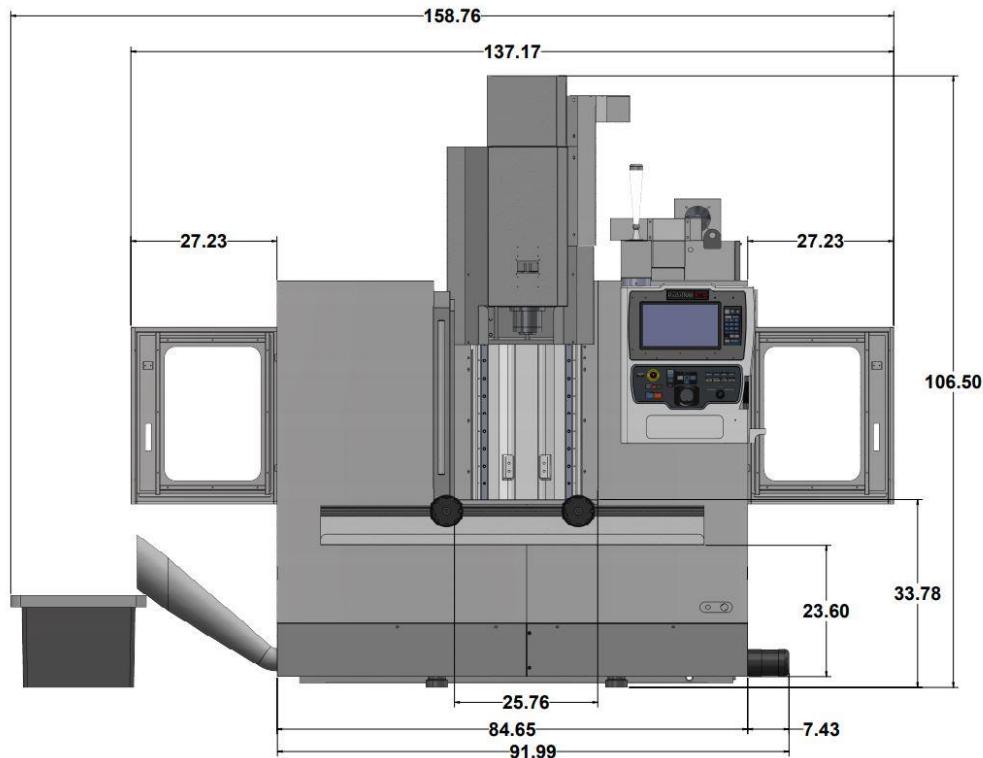
Option Key
Each ProtoTRAK machine will come with a USB Option Key that will be installed inside of the pendant. This key must remain plugged in to allow certain standard software features to work and also to allow optional software features to work.
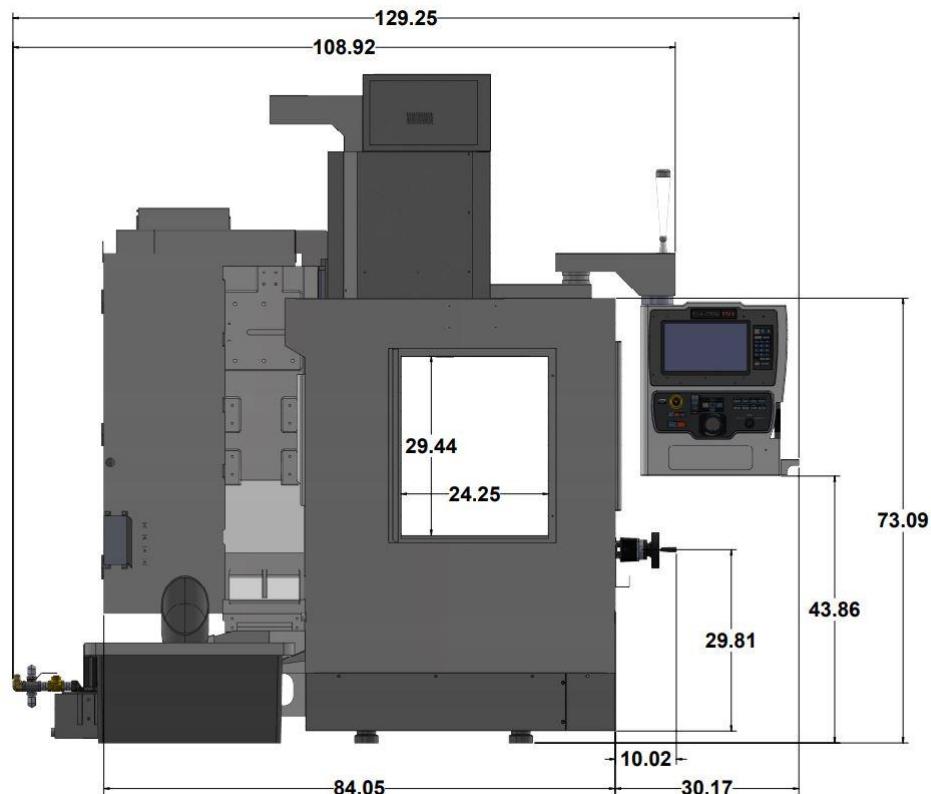
3.4 Machine Details
3.4.1 Physical Layout and Dimensions

VMC2 Front View
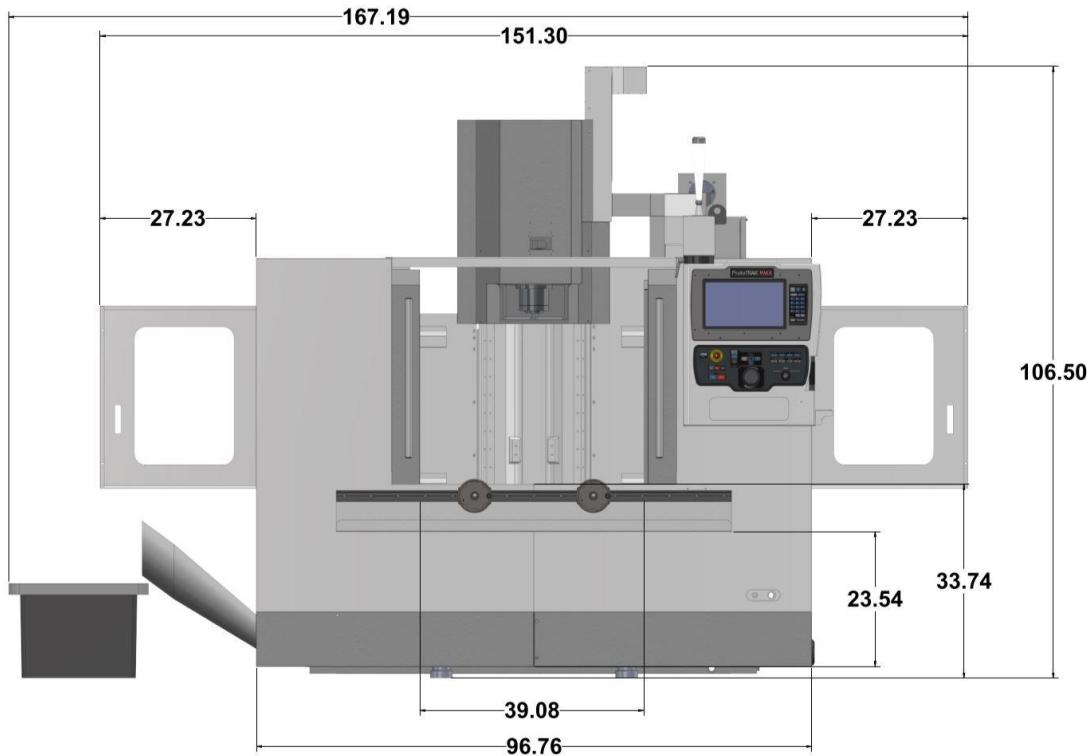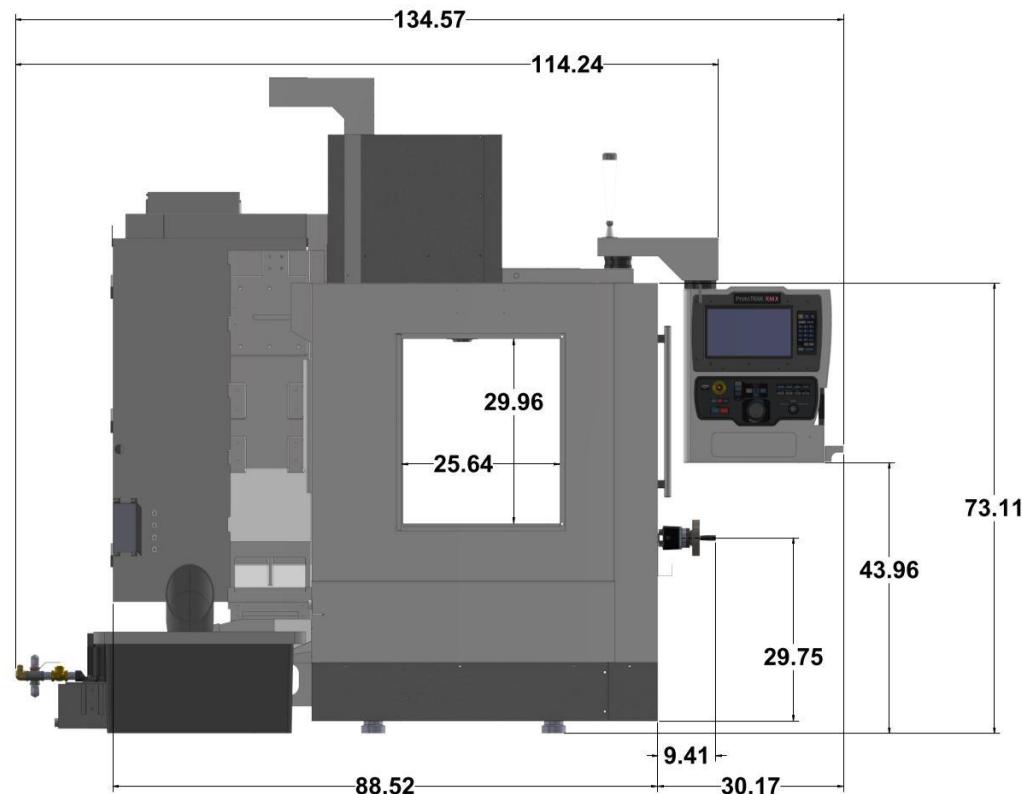
VMC2 Left Side View
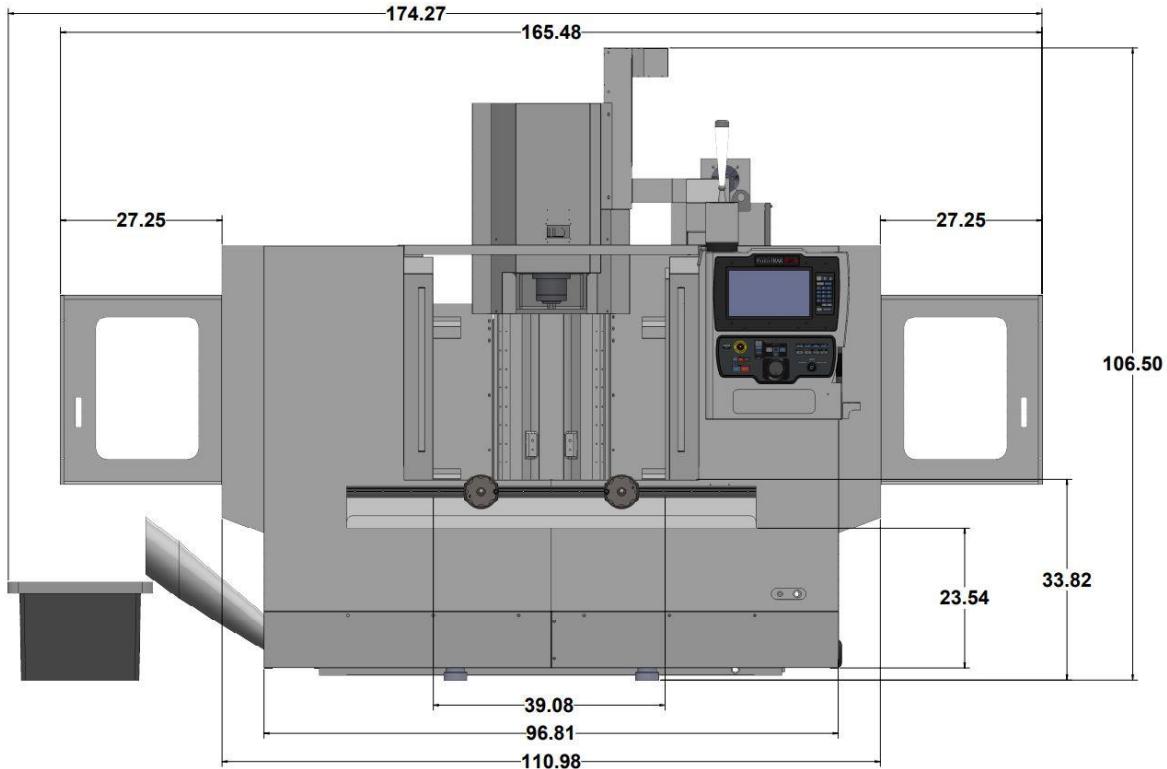
TMC5 Front View

TMC5 Left Side View
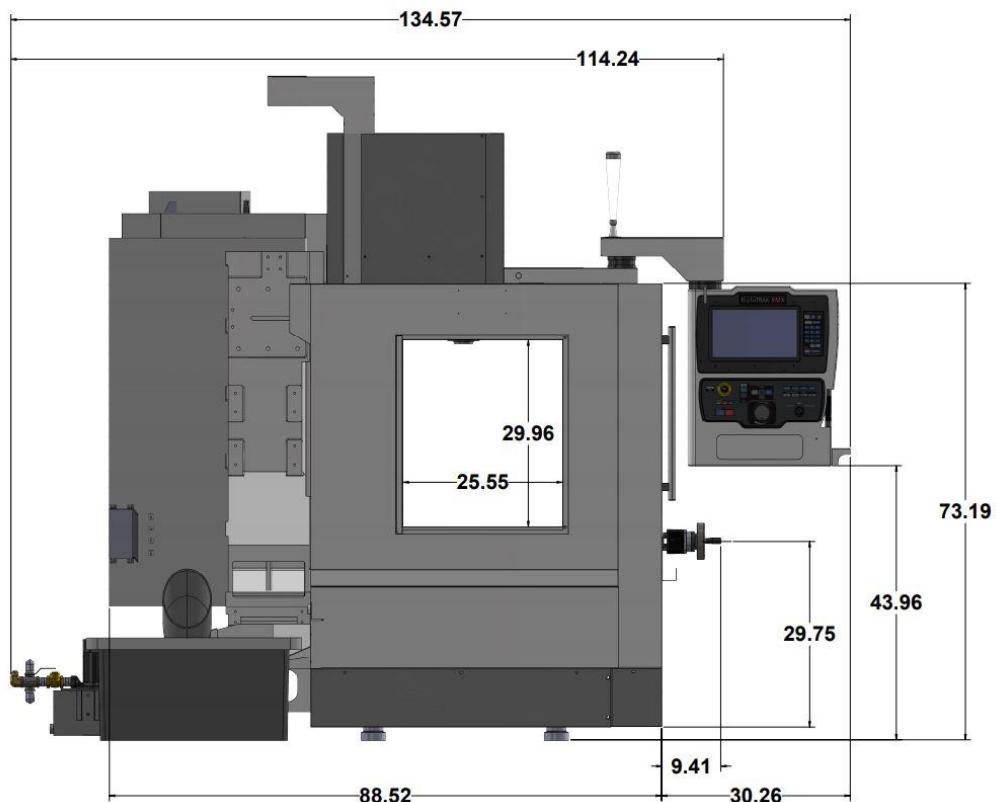
TMC7 Front View
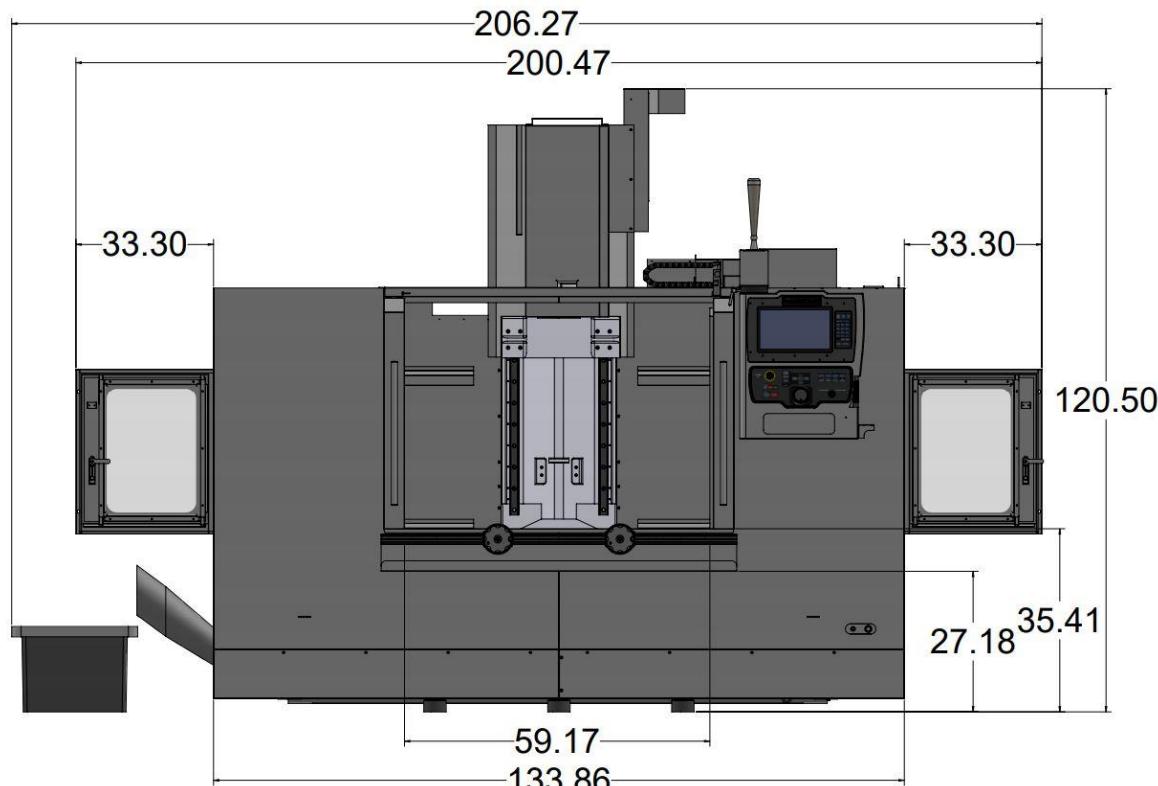
TMC7 Left Side View
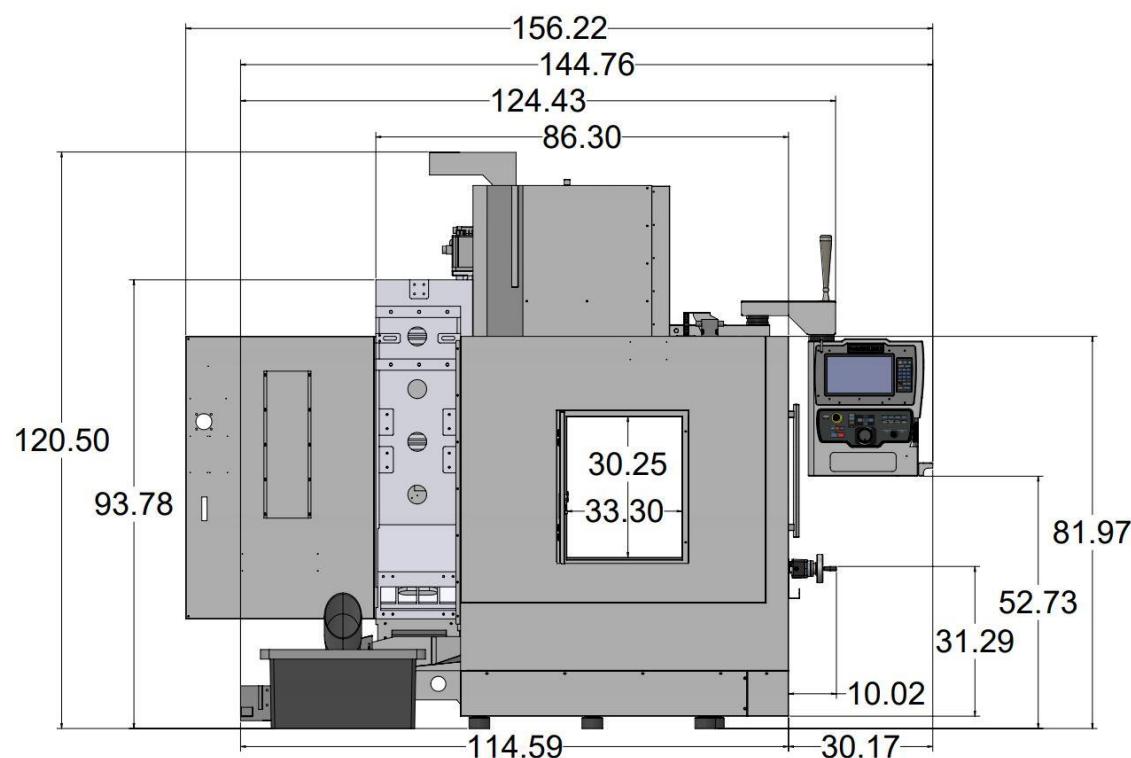
TMC10 Front View

TMC10 Left Side View

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
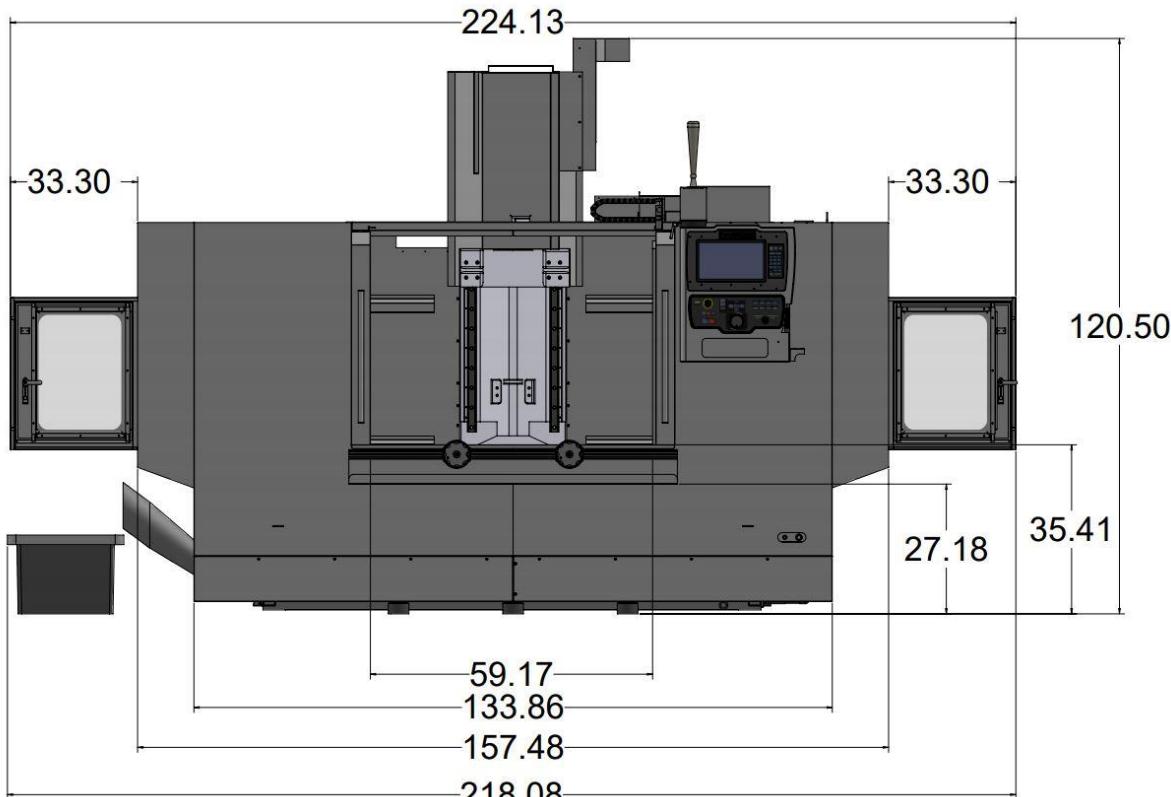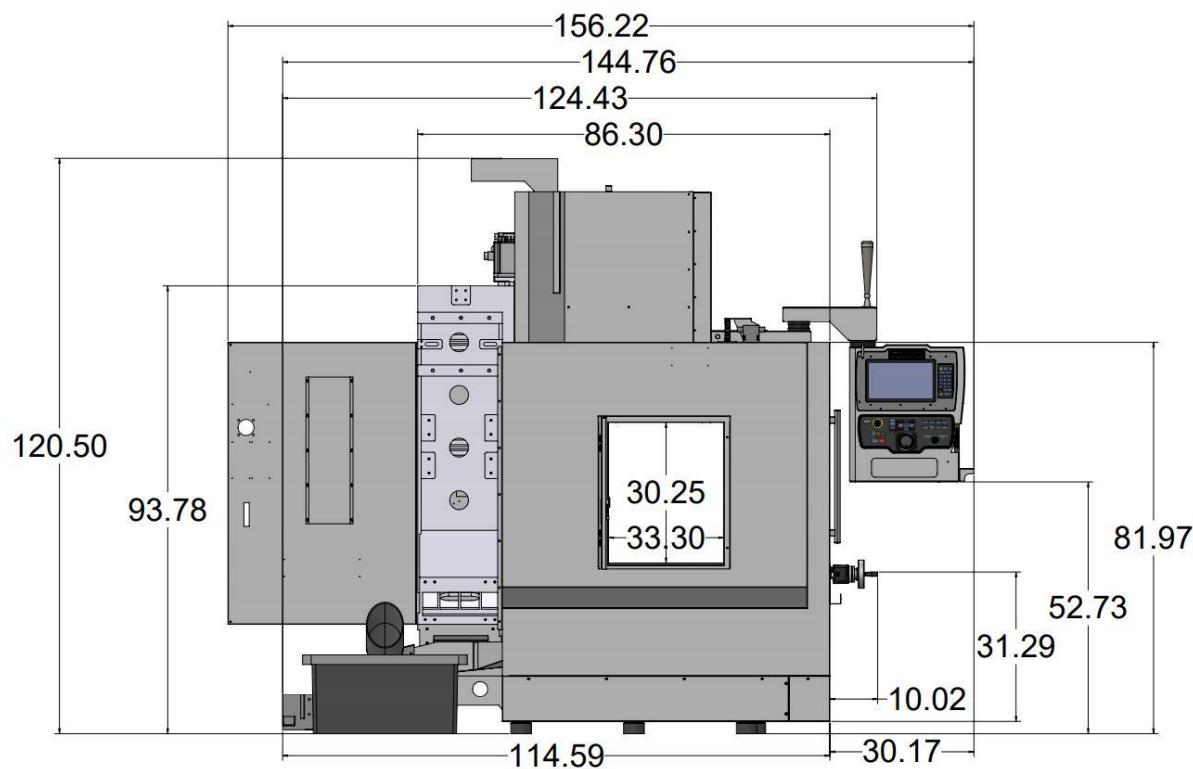
TMC12 Front View

TMC12 Left Side View

TMC14 Front View

TMC14 Left Side View

3.4.2   Machine Specifications

MODEL NAME	VMC2	TMC5	TMC7	TMC10
Table Size	18" x 15"	27.56" x 15.75"	35.43" x 19.69"	44.09" x 19.69"
T-Slots (number x width x pitch)	4 x 16 x 63 mm	3 x 18 x 100 mm 3 x .709" x 3.937"	5 x 18 x 100 mm 5 x .709" x 3.937"	5 x 18 x 100 mm 5 x .709" x 3.937"
Travel (X, Y, Z axis) with soft limits	14" x 12" x 17"	20" x 16" x 20"	30" x 20" x 20"	40.75" x 20" x 20"
Travel (X, Y, Z axis) with no soft limits *User may cause damage if they crash. Travel distance is just shy of hard stops.	14.8" x 18.8" x 17.8"	21.25" x 17.50" x 21.25"	30.75" x 21.25" x 21.25"	42" x 21.25" x 21.25"
Spindle Taper	30 Taper	40 Taper		
Spindle Speed Range – Production Ops	50-10,000	50-8,000		
Spindle Speed Range – Production Ops (12K Spindle Option)	N/A	50-12,000		
Spindle Speed Range – Tool Room Ops	50-5000			
Spindle Nose Diameter	2.2"	3"		
Tool Clamping Force (90 psi)	850-1000 lbs.	1500 lbs.		
Tool Holder Type	BT30	CAT 40 or BT40		
ATC Tool Capacity	8	16		
Maximum Tool Weight Incl. Holder	8 lbs.	15 lbs.		
Maximum Tool Diameter	2"	3.14"		
Carousel Speed	N/A	.8 sec from station to station		
Tool Carrier Speed	1.2 – 1.4 sec from front to back	N/A		
Tool Selection System	Shortest Path	Bi-directional/Shortest Path		
Retention Knob	See drawing of retention knob			
Spindle Motor Power (continuous)	3 HP	10 HP		
Spindle Motor Power (peak)	4.5 HP	15HP		
Power requirements – 208V (volts, phase, current) 8k Spindle	208V (208-240V is acceptable), 3P, 30A	208V (200-240V is acceptable), 3P, 68A (Current can be as high as 87 amps with all options ordered – CTS (+14 amps), 4th axis (+5 amps))		
Power requirements – 208V (volts, phase, current) 12k Spindle	208V (208-240V is acceptable), 3P, 30A	208V (200-240V is acceptable), 3P, 69A (Current can be as high as 88 amps with all options ordered – CTS (+14 amps), 4th axis (+5 amps))		
Power requirements – 480V (volts, phase, current) 12k Spindle	480V w/ transformer option (440 to 520 V is acceptable), 3P, 15A	480V with transformer option (432-528V is acceptable), 3P, 46A (46 amps-based nominal rated current of transformer)		
Power requirements – 480V (volts, phase, current) 12k Spindle	480V w/ transformer option (440 to 520 V is acceptable), 3P, 15A	480V with transformer option (432-528V is acceptable), 3P, 46A (46 amps-based nominal rated current of transformer)		
Maximum Weight of Workpiece	500 lbs	1000 lbs.		
Height of Table from Bottom of Bed***	34"	38"	38"	38"

MODEL NAME	VMC2	TMC5	TMC7	TMC10
Minimum Spindle Nose to Table Distance****	2.75"	3.5"	3.5"	3.5"
Maximum Spindle Nose to Table Distance****	20.25"	23.5"	23.5"	23.5"
Distance of Tool Carousel to Table	N/A	18"	17.5"	17.5"
Spindle Center to Spindle Head Casting Face	15.7"	16.50"	18.25"	18.25"
Minimum Machine Height***	93"	90"	90"	90"
Maximum Machine Height (head all the way up)***	101.5"	106.5"	106.5"	106.5"
Minimum Height to Fit Machines Through Doorway**	93"	84"	84"	84"
Length (pendant enclosure rotated at 90º and coolant tank in place)	88.25"	129.25"	135"	135"
Overall Width of Machine with Auger Motor Installed	N/A	92"	97"	111"
Overall Width of Machine with Side Doors Open and Auger Chute and Chip Container in Place	N/A	159"	167.25"	174.50"
Overall Length of Machine with Electrical Cabinet Full Open 180º	113.5"	146"	152"	152"
Footprint of Machine (width x length) (coolant tank in place)	55" x 113.5"	92" x 129.25"	97" x 135"	111" x 135"
Weight Net / Shipping (lbs.)	2,825 / 3,175	7,260 / 7,610	8,360 / 8,710	9,900 / 10,250
Rapid Traverse X, Y, Z	400 ipm in Tool Room Ops 600 ipm in Prod Ops	400 ipm in Tool Room Ops 1000 ipm in Production Ops		
Coolant Capacity	15 gallons	56 gallons		
Coolant Pump	¼ HP	750 Watt		
Wash Power	N/A	750 Watt		
Air - Pressure CFM or SCFM	2.5 CFM at 90 psi or 18 SCFM			
Air Quality	Air dried/ filtered water separator upstream of the Machine			
Lube Pump Capacity	2 liters			
CTS (Coolant Through Spindle) Power	N/A	3 KW		
CTS Max Pressure	N/A	319 psi		

* Specs subject to change.

** Must remove resistor and all sheet metal attached to top of electrical cabinet. The cable way for Z axis also needs to be removed. The lifting point on the electrical cabinet is now the highest point.

*** Height changes slightly based on how machine is leveled on leveling screws.

**** Can vary slightly due to home switch location and soft limit settings.

MODEL NAME	TMC12	TMC14
Table Size	23.62" x 51.18" 600 x 1300 mm	23.62" x 62.99" 600 x 1600 mm
T-Slots (number x width x pitch)	5 x 18 x 125 mm 5 x .709" x 4.921"	5 x 18 x 125 mm 5 x .709" x 4.921"
Travel (X, Y, Z axis) with soft limits	50" x 27" x 25"	60" x 27" x 25"

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

MODEL NAME	TMC12	TMC14
Travel (X, Y, Z axis) with no soft limits *User may cause damage if they crash. Travel distance is just shy of hard stops.	~51.28" x 27.66" x 26.12"	~61" x 27.66" x 26.12"
Spindle Taper	40 Taper	
Spindle Speed Range – 8K	50 – 8000	
Spindle Motor to Spindle Pulley Ratio – 8K	1 to 1	
Spindle Speed Range – 12K	50 – 12,000	
Spindle Motor to Spindle Pulley Ratio – 12K	1 to 1.25	
Spindle Nose Diameter	3 ¾"	
Diameter of Spindle	70 mm	
# of Spindle Bearings	4 total – 2 lower and 2 upper	
Spindle Bearing Sizes	110 x 70 x 20 mm	
Tool Clamping Force (90 psi)	2200 lbs.	
Tool Holder Type	CAT 40 or BT40	
ATC Tool Capacity	16	
Maximum Tool Weight Incl. Holder	15 lbs.	
Maximum Tool Diameter	3.14"	
Tool Selection System	Bi-directional / Shortest path	
Retention Knob	See drawing of retention knob	
Spindle Motor Power (continuous) – 8K/12K	15 HP	
Spindle Motor Power (peak) – 8K/12K	22.5 HP	
Spindle Motor Torque (continuous) – 8K	43.77 ft-lbs	
Spindle Motor Torque (peak) – 8K	65.66 ft-lbs	
Spindle Motor Torque (continuous) – 12K	35.01 ft-lbs	
Spindle Motor Torque (peak) – 12K	52.52 ft-lbs	
# of X, Y and Z axis linear bearing blocks	3 on each guide on X axis 2 on each guide on Y and Z axis	
X, Y and Z axis linear guide size	45 mm	
X axis linear guide way span	380 mm	
Y axis linear guide way span	900 mm	
Z axis linear guide way span	440 mm	
X, Y and Z axis ballscrew diameter	40 mm	
Pitch of ballscrew (X, Y and Z axis)	10 mm	
Ballscrew accuracy class	C3	
Rapid traverse X, Y, Z	1000 ipm	
Cutting max speed X, Y, Z	1000 ipm	
Power Requirements – 208V (volts, phase, current) 8k spindle	208V (200-240V is acceptable), 3P, 93A *(Current can be as high as 112 amps with all options ordered - CTS (+14A), 4th axis (+5A))*	
Power Requirements – 208V (volts, phase, current) 12k spindle	208V (200-240V is acceptable), 3P, 96A *(Current can be as high as 115 amps with all options ordered - CTS (+14A), 4th axis (+5A))*	
Power Requirement – 480V (volts, phase, current) 8k spindle	480V w/ transformer option (432-528V is acceptable), 3P, 49A *(Current can be as high as 58 amps with all options ordered - CTS (+6A), 4th axis (+3A))*	

MODEL NAME	TMC12	TMC14
Power Requirement – 480V (volts, phase, current) 12k spindle	480V w/ transformer option (432-528V is acceptable), 3P, 49A *(Current can be as high as 58 amps with all options ordered - CTS (+6A), 4th axis (+3A))*	
Maximum Weight of Workpiece	1,000 Kg / 2,200 lbs.	1,200 Kg / 2,640 lbs.
Height of Table from Bottom of Bed***	38.37" 1000 mm	
Minimum Spindle Nose to Table Distance****	2.25" 57 mm	
Maximum Spindle Nose to Table Distance****	28.75" 735 mm	
Distance of Tool Carousel to Table	21.5"	
Spindle Center to Spindle Head Casting Face	27.56" 700 mm	
Minimum Machine Height***	105"	
Maximum Machine Height (head all the way up)***	120.5"	
Minimum Height to Fit TMC's Through Doorway (Remove Z motor and sheet metal on top of column)	93"	
Length (pendant enclosure rotated at 90º and coolant tank in place)	144.76"	
Overall Width of Machine with No Auger Chute, Auger Motor or Chip Container	133.86"	157.48"
Overall Width of Machine with Side Doors Open and Auger Chute and Chip Container in Place	206.27"	224.13"
Overall Length of Machine with Electrical Cabinet Full Open 180º	156.22"	
Footprint of Machine (width x length) (coolant tank in place)	133.86" x 144.76"	157.48" x 144.76"
Weight Net	14,300 lbs. / 6500 kg	16,500 lbs. / 7500 kg
Shipping Weight	14,800 lbs. / 6727 kg	17,000 lbs. / 7727 kg
Air - Pressure CFM or SCFM	3.0 CFM (at 90 psi) or 25 SCFM	
Air Quality	Air dried / water separator upstream of the TMC	
Lube Pump Capacity	2 liters	
Lube Oil	ISO 32 or 10 W	
Coolant Pump (watts)	750	
Coolant Pump Pressure and Volume	32.8 liters/min at 3 kg/cm2 8.7 gpm at 42 psi	
Coolant Wash Pump (watts)	1290	
Coolant Wash Pressure and Volume	66.3 liters/min at 3 kg/cm2 17.5 gpm at 42 psi	
CTS Coolant Through Spindle Pump (watts) (option)	3 KW 4.02 HP	

MODEL NAME	TMC12	TMC14
CTS Pressure	22.4 kg/cm2 or 2200 Kpa 319 psi	
Tank Capacity	267 liters 71 gallons	

* Specs subject to change.

*** Height changes slightly based on how machine is leveled on leveling screws.

**** Can vary slightly due to home switch location and soft limit settings.

3.5   Optional Equipment
3.5.1   Electronic Handwheels
Electronic handwheels are optional only on the TMC's. When ordered as part of the TRAKing / EHW Option, the electronic handwheels will operate when the ProtoTRAK RMX CNC is in a Mode where the machinist controls the motion of the table and saddle. This includes the DRO Mode, the Set-Up Mode and the TRAKing operation in the Run Mode. Electronic handwheels will not operate during other functions, such as edit or program in/out mode.

While in TRAKing, the X handwheel is course resolution and the Y handwheel is fine resolution. The F/C key on the run panel does not come into play here.

3.5.2   Tooling Measurement Cart
The Tooling Measurement Cart is optional on the VMC2 and TMC's and consists of the Tooling Cart, Preset Tool, and Plate. The Tooling Measurement Cart helps set tool offsets in the tool table without touching tools off the part. This is very handy when the machine has Electronic Hand Wheels and the chances of damaging the part are high.

To use, set a base tool using Tooling Cart, and then for all other tools load it in the tooling cart read the value from the measurement scale and enter the absolute value as tool offset for that tool in tool table.
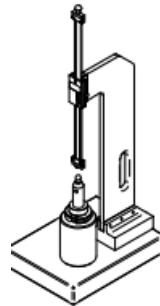
Figure 3.5.2 - Preset Tool & Plate

3.5.3   Auxiliary Functions
Auxiliary functions are controlled through the ProtoTRAK RMX CNC either in the program or with the keys on the run panel. The Auxiliary functions consist of the following:
- Coolant
- Air
- Pulse Indexer *(Optional)*
- Part Change Position
- Programmable Auxiliary Output to control secondary external device *(Optional)*
- Coolant through Spindle (CTS)

3.5.4   Internal 256GB Memory
SSD for program storage with at least 256 GB of storage memory. The SSD will be located inside the programming and run panel enclosure. Installation instructions will be included in the kit if ordered.

3.5.5   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.

3.5.6 Gripper BT40 Tooling

If a user wants to use BT40 Tooling, this option must be ordered to replace the 16 grippers in the ATC with BT40 grippers. (***Note –*** *Available only for the TMC's.*)

3.5.7 Gripper BT30 Tooling

A package of tool holders to fit the 30-taper spindle. (***Note*** *– Available only for the VMC2.*)

Includes:

- 2 each end mill holder ½"
- 2 each end mill holder 3/8"
- 3 each ER-16 collet holder
- 1 each shell mill holder

3.5.8 ProtoTRAK RMX Offline/RMX Additional Seat

Optional piece of software you can purchase to program your ProtoTRAK on your desktop computer.

3.5.9 Retention Knobs

The VMC2 uses BT30 retention knobs while the TMC's use CAT40 retention knobs, as shown below. If the CTS option is purchased for the TMC's, you will need retention knobs similar to figure 3.5.9b but with holes. Tightening to the proper torque value is important for all retention knobs. Please see the retention knob manufacturer for the proper torque.
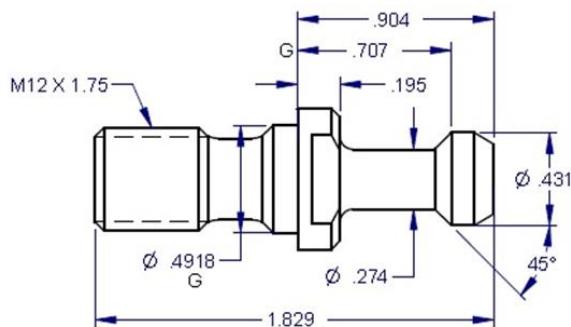
Figure 3.5.9a - VMC2 Retention Knobs   ***Figure 3.5.9b - TMC Retention Knobs***
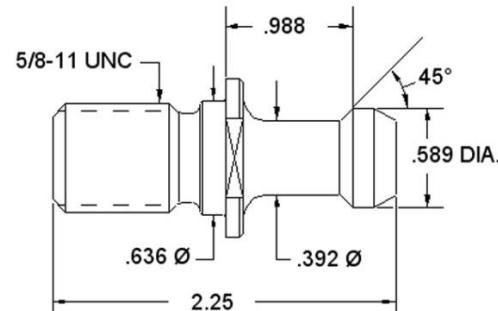
Retention knobs can be purchased through TRAK Machine Tools under part numbers:

- RETN-KNOB-VMC2 (***Note*** *- Only for the VMC2.*)
- RETN-KNOB-KIT (***Note*** *- Only for the TMC's* without *the CTS option added.*)
- RETN KNOB KIT-CTS (***Note*** *- Only for the TMC's* with *the CTS option added.*)

Retention knobs purchased through TRAK Machine Tools should be torqued to 30 ft-lbs.

WARNING!
Retention knobs come in a wide variety of designs; however, they often look similar and appear to be interchangeable, but they are not. Use only the knob that the machine is designed to use. The use of the incorrect knob, or the incorrect use of a knob, may result in injury and/or damage to the mechanism.

3.5.10 Step Down Transformer

The TRAK VMC2 and TMC's run on 200 to 240-volt 3 phase power.

TRAK VMC2:

Shops with 440 to 520-volt incoming power will require an optional step-down transformer for the VMC2.

The TRAK VMC2 mill must be ordered for 200 to 240 or 440 to 520 volts. 200 to 240V machine does not require a transformer. For 440 to 520V machine, the factory input can be connected to one of the three input taps, viz. 440V, 480V or 520V. The transformer output is set to 220V.

TRAK TMC's:

Shops with 415 to 480-volt incoming power will require an optional step-down transformer for the TMC's. The factory input can be connected to one of the three taps, viz. 400V, 440V or 480V. The transformer output can be set to either 200V or 220V.

WARNING!
The input voltage to the machine should not exceed 240 or 520 volts. The spindle inverter is not rated for voltages higher than this. If the shop has voltages that exceed these numbers, they will need a step down transformer to lower the voltage. Voltages higher than this may void your warranty.

3.5.11 USB Memory Option

We highly recommend that you use the high-quality Delkin® brand thumb drive we provide with the USB Memory Option.

3.5.12 Vise

A 6" vise from Kurt is available. We supply their DX6 mode. The vise kit comes with all the necessary hardware to fasten it to your table.

3.5.13 Pallet Jack

The optional pallet jack for the VMC2 is a narrow pallet jack with forks about 20" apart which can be used on all four sides. Standard pallet jacks with approximately 27" forks will not lift the VMC2 mill safely from the front or rear and not align with lift points on either side of the mill.

Note - *Available only for the VMC2.*

3.5.14 4th Axis Option

The 4th axis option is only available for the TMC's and can be retro-fitted at any time after its initial purchase. Once the option is purchased and physically installed, you can activate it on your software screen. See section 15 for additional information on the 4th Axis.

3.5.15 12K Spindle Option

All TMC machines come standard with an 8K spindle with an option to upgrade to a 12k spindle for increased spindle speed. This allows the spindle to run from 50 RPM up to 12,000 RPM. Once the option has been purchased and physically installed, you can begin using it.

3.5.16 CTS 8K & CTS 12K Option

Optional coolant available for the TMC's which runs through the spindle to support higher cutting speeds. Benefits of the through spindle coolant include:

- Eliminating peck cycles in most deep hole drilling and increase both, speeds and feeds.
- Enabled use of higher quality carbide tools that produce better overall quality.
- Improved throughput by reduced cycle times.
- Clears chips easily and speeds up the cutting process to produce improved surface finishes.

Note - *The CTS 8K Option is available for TMC's that come with the standard 8K Spindle, and the CTS 12K option, is for TMC's that are upgraded to the 12K Spindle option.*

3.6 Standard Equipment
3.6.1 Lubrication System

The automatic lubricating system is a centralized system. The way and ballscrew lubrication pump is wired to operate when the spindle is running. It also turns on each time you turn on your machine to lubricate the axis before movement.

Factory Default Values
Interval Time – 30 min.
Discharge Time – 5 sec
Discharge Pressure – Approximately 100 – 150psi

While the system is automatic, it is recommended that after long idle periods, the machine be manually lubricated by using service code 300. Cycle the lube pump two to three times. The lubrication reservoir should be maintained on a daily basis, filling only with high quality lubricating oil. The control monitors if the lube pump oil level is low or the pressure is low during operation and will warn the user if either condition exists. See the service manual for more information.

CAUTION!
Failure to properly lubricate the mill will result in the premature failure of bearings and sliding surfaces.
Always Observe Low Air Pressure and Low Oil Level Warnings.

3.6.2 Servo Motors

VMC2: Servo motors on table, saddle are 750-watt brushless motors. The motor driving the z-axis is a 1-kilowatt brushless motor.

TMC's: Servo motors on table, saddle are 1-kilowatt brushless motors. The motor driving the z-axis is a 2-kilowatt brushless motor.

3.6.3 Reference (Base) Pin

Each TRAK VMC2 and TMC is shipped with a 3/8" shank pin that is meant to make tool setting more convenient. You will find the pin in the Hardware Kit that accompanies the machine. The idea is for you to use this pin and one of your tool holders to make a Base Reference Tool. That way, you can use the tool as your Base Tool when you set up a new Z Offset on a tool.

We recommend that you keep the pin in the tool holder as a permanent part of your toolbox. You will find that that time you save and the peace of mind you get from having this secure reference is well worth the dedication of a tool holder.

3.6.4   Status Lights

There is a light attached to the top of the machine to provide the user with a status of what is occurring at that moment.

The lights perform as follows:
- The green light is illuminated when the machine is running a program.
- The yellow light is illuminated when operator input is required, like when a part change or a manual tool change needs to be done, and if the program has stopped running.
- The red light is illuminated when an alarm has occurred.

3.6.5   Electrical Cabinet

The electrical cabinet is found at the rear of the machine, on the right side. The electrical cabinet contains the main control hardware for the machine. The main components consist of: computer module, AC spindle drive, servo drives, relays, and contractors.

3.6.6   Door Lock Logic

There is a Door Lock LED button located on the run panel which when turned ON enables auto lock logic and when turned off disables the auto lock. When turned ON it allows the user to stop keeping track if the door is locked or not as the ProtoTRAK TMC locks the door throughout necessary functions such as: during running a program, tool change, homing, ATC moving. The ProtoTRAK also unlocks the door automatically when the Door Lock LED is ON for example, when there is a manual tool change or when program RUN is over. The door can be unlocked anytime by pressing the Door Lock LED OFF.

By default, during tool changes the door will automatically lock and unlock irrespective if the led for Door Lock is ON or OFF.
Note - *Not available on the VMC2.*

3.6.7   Chip Removal System

Comes standard on TMC's. The TMC uses an auger chip removal system. When the forward direction is chosen on the run panel, the auger will displace chips into the chip cart. It can be run momentarily in the reverse direction in order to free a jam.
Note - *Not available on the VMC2.*

3.7 Standard Equipment – Applicable Only on the VMC2
3.7.1 Floor Connection
The TRAK VMC2 has 4 adjustable leveling feet. It is important when moving the machine from one spot to another that all 4 feet are touching the floor. As floors vary, some adjustment may be necessary as you move the machine from one place to another.

For work that requires a high level of precision, you may have to level the machine using precision levels. In addition to this, the user should check the tram of the spindle relative to the table and adjust one or more of the 4 corners to adjust the tram within 0.001" TIR.

3.7.2 Power Cord
The TRAK VMC2 comes with a 208 or 480-volt power cord/plug so the machine can be easily moved around the shop. If the single-phase option is purchased, the machine will then come with a single-phase plug.

The female connector that needs to be provided by the end user should match the connector that is provided. See figure below for images of the plugs. From left to right they are as follows:
1. 208-volt 3-phase 30-amp plug
2. 480-volt 3-phase 20-amp plug
3. 208-volt single phase 30-amp plug

Figure 3.7.2 - Available Power Plugs

3.7.3 Coolant and Coolant Wash System
The coolant and coolant wash system uses one pump to provide coolant to the work and also to wash chips away from the area where the ATC comes out. Wash areas can be controlled by the flexible coolant lines found at the base of the enclosure.

The coolant tank holds approximately 15 gallons of coolant.

3.7.4 Pneumatic System
The machine requires a supply of compressed air between 85-100 psi with a recommended air supply of ½" I.D, minimum is 3/8" ID. Air pressure to pneumatic components, the ATC slide mechanism, and air purge (internal spindle) can be controlled individually by means of the adjusting valves located at the back of the TRAK VMC2 mill.

3.7.5 Enclosure Doors

The front door has an electro-mechanical safety interlock that must be engaged when running a CNC program. If the door is opened during a machining operation, the program will be shut down.

The enclosure is also equipped with left and right doors that are bolted in place.

3.7.6 Chip Removal

The chip pan is located in the front of the VMC2. To remove, lift up on the pan and pull forward.

4.0 Basic Operation

The TRAK Machining Centers have been designed to give you unprecedented power in a CNC user experience that is natural and easy to use. This section will walk you through some basics of using the ProtoTRAK RMX CNC.

4.1 Powering Up the System

To turn on power to the entire machine and control system, rotate the power switch on the Electrical Cabinet to the ON position.

The control will go through its boot up sequence. When it is finished, the following screen will appear.
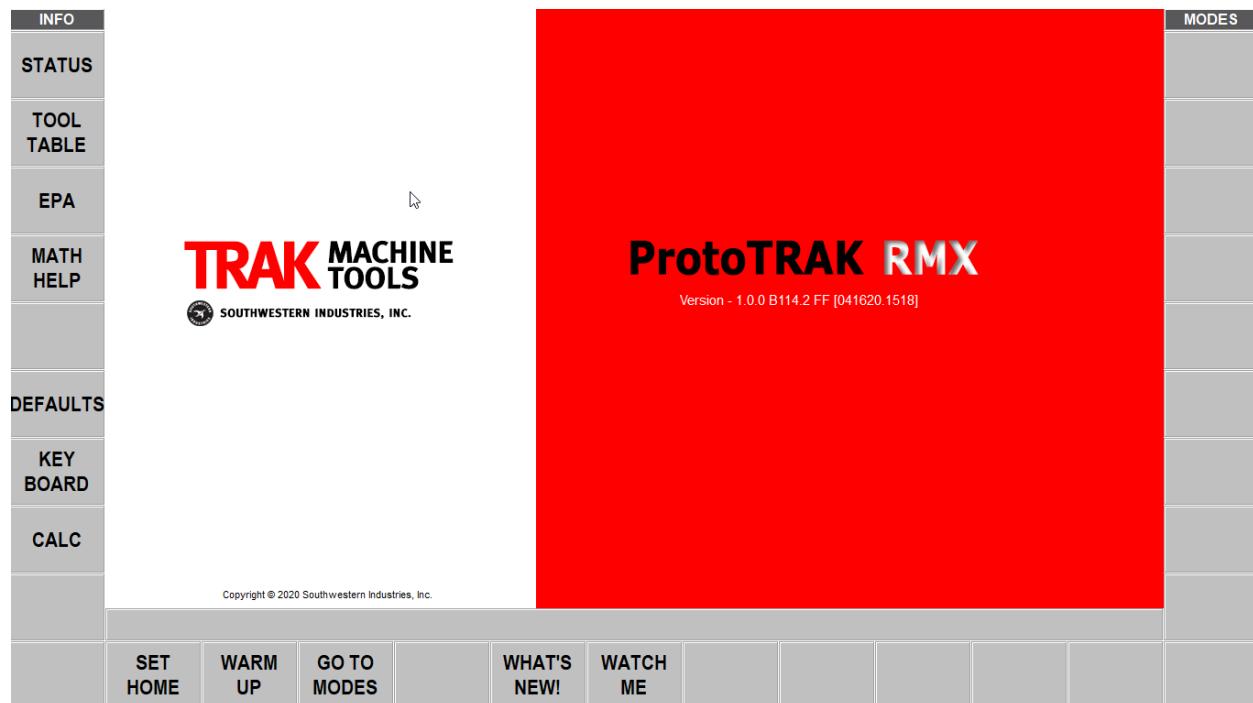
Figure 4.1 - The ProtoTRAK RMX will display this screen after it boots up.

To begin using the ProtoTRAK RMX CNC, first do the following:

Press the **POWER/RESET** hard key and hold it for a few seconds until the white LED comes on.

Select the **SET HOME** soft key.

Tap the **WARM UP** soft key and select either 10 minute or 20-minute warm up. Once the warm up is done, you will return to the screen as shown in figure 4.1.

Select the **GO TO MODES** soft key.

4.2   Shutting Down the ProtoTRAK RMX CNC

IMPORTANT
The system must be turned off properly or you may lose unsaved data such as programs or certain machine configurations. The SHUT DOWN soft key will initiate the process of shutting down safely. When the screen goes blank, turn the power switch at the back of the electrical cabinet to the OFF position.

In order to reach the SHUT DOWN soft key, first close any Fly Out Windows that are open. We also recommend that you complete any prompted activity, such as programming an Event. Press the BACK hard key repeatedly until you get to the screen shown in Figure 4.2 below.
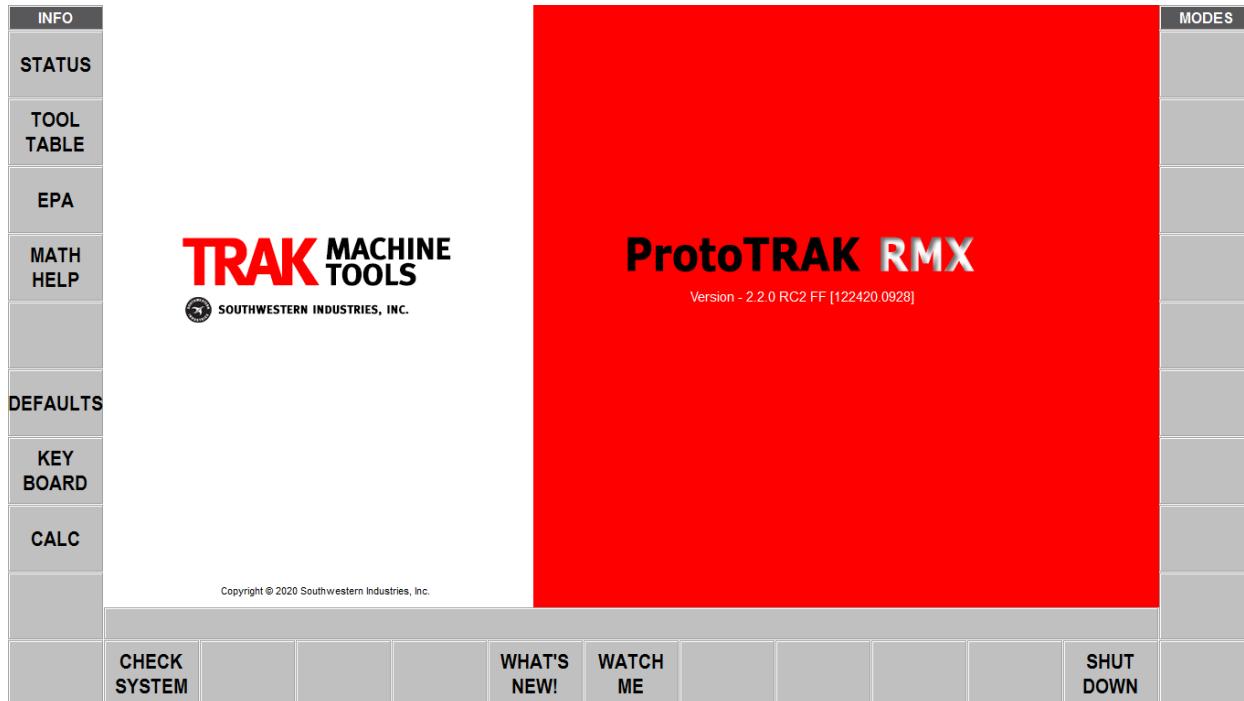
Figure 4.2 - The ProtoTRAK RMX will display this screen once you've reached the SHUT DOWN soft key.

The ProtoTRAK control should be shut off at a minimum every few days. Failure to reboot may cause the control to run slowly. For example, the INFO screens may move in and out slowly. This is a sign the control should be restarted. Take advantage of the SAVE TEMP feature prior to shut down as mentioned in the note below.

Note - *You can save your current positions, tool information and program using the Save Temp feature in the Program In/Out mode.*

4.3 Operator Run Keys

4.3.1 Spindle Control

The Spindle Forward/Off/Reverse is controlled through the Spindle hard keys located on the ProtoTRAK RMX control pendant run panel. The three buttons on the front control, FORWARD, STOP, REVERSE, will operate the spindle in DRO and Run mode.

The spindle will not operate unless the ProtoTRAK CNC is in DRO or Run Mode.

In Toolroom Ops, the user needs to turn the spindle on. In Production Ops, the spindle automatically starts in run mode.

4.3.2 Feed

The Feed Keys control the Go or Stop of programmed toolpath in the Run Mode. When running a program in the Run Mode, you will be prompted for when to start the feed by pressing the GO button.

WARNING!
Pressing the GO button will initiate automatic machine motion!

4.3.3 Front Panel EHW

The center EHW button turns on the Front Panel Electronic Handwheel and the operator can then choose which axis to move. Once an axis is selected, you can select any of the four speed options. Available speed options are 0.0001"/0.002 mm, 0.0010"/0.020mm, 0.0100"/0.200mm, and in fast axis move up to 300 ipm. Rotate the knob located below the keys to its desired position.

4.4 Manual Operation of the Ram, Table, and Saddle

Using the electronic handwheel(s), the TRAK Machining Centers may be used manually while in the DRO Mode. Manual positioning and jog are active in other situations when needed, such as when setting tools.

When not in the DRO Mode, moving the Electronic Handwheel(s) will not result in movement of the machine.

4.5 Data Input Keys

Data Input keys (hard keys) are used to input numeric date and set as an Absolute or Incremental reference.

4.6 Modes

The Modes soft keys are always active. With the ProtoTRAK RMX, it is always possible to switch freely between Modes once you close any Fly Out window.

As you move into and out of a Mode, the operations will almost always be where you left them the last time you went out.

This manual will deal with each Mode thoroughly in the Sections following.

4.7   Gestures

The touchscreen is operated with the following gestures:

- **Tap:** Tap is used to select an item such as an Info Key. To Tap, lightly touch the screen with one finger.

- **Double Tap:** In Program In/Out mode, double tapping will open a file or folder.

- **Swipe:** Swipe is used to move from one screen to another, for example you move forward and back through Events in the Program Mode with swipes. To Swipe, touch the screen with one finger and move rapidly in the direction you want to move.

- **Pan:** Pan is used to move content, such as a part drawing, around the screen. To Pan, touch the screen with one finger and move slowly in any direction.

- **Spread / Pinch:** These gestures are used to zoom in or out of a drawing. Use your thumb and finger.

- **Touch and Hold:** Touch and hold works much like the right click of a mouse to initiate a pop up menu for selections, when one is available. Touch the screen item and hold for 2 seconds.

- **Drag:** Drag is used to move some of the information windows to more convenient location on the screen. Touch the window header or title area and drag it to where you want.

- **Rotate:** Rotate is used to rotate a drawing or a solid model. Touch the screen with 2 fingers and twist to rotate the part along and axis projecting out of the screen. Move your 2 fingers to the right to rotate around a vertical line on the screen. Move your 2 fingers to the left to rotate around a horizontal line on the screen. Move your 2 fingers CW or CCW on the screen to rotate around a point.

5.0 Definitions, Terms, and Concepts

5.1 ProtoTRAK RMX CNC 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.
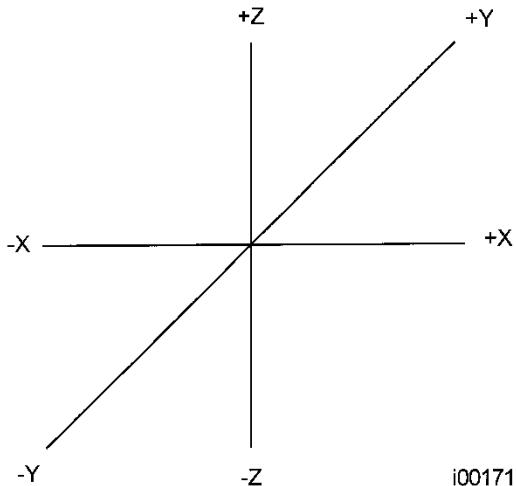
Figure 5.1 - ProtoTRAK RMX CNC Conventions

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 and Tool Path Programming

The ProtoTRAK RMX CNC gives you ultimate flexibility in programming. Programs that are entered through the ProtoTRAK RMX CNC 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 RMX CNC 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 places

Using Geometry Programming, it is impossible for the ProtoTRAK RMX CNC 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.

Programs generated by CAD/CAM systems are always generated as Tool Path programs and are run as such even if the Advanced Features with Verify is not active on the ProtoTRAK RMX CNC.

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 RMX CNC can machine arcs in any vertical plane rather than just XZ or YZ.
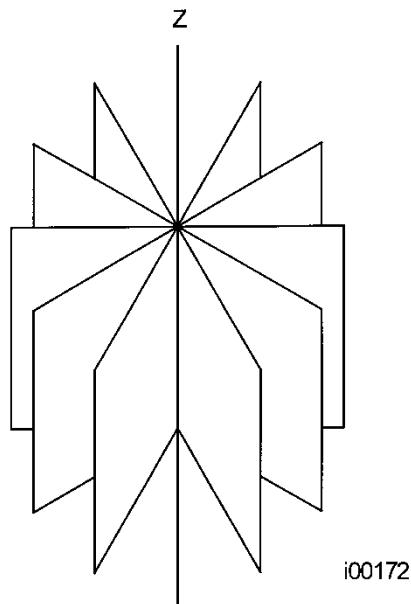
Figure 5.3 - Vertical Planes

5.4 Absolute and Incremental Reference

The ProtoTRAK RMX CNC 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:
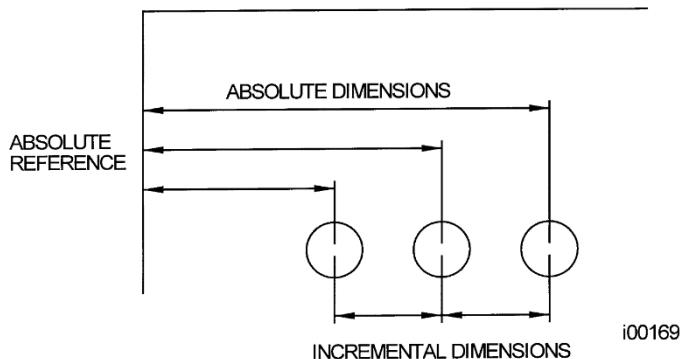
Figure 5.4 - Each point has both an absolute and an incremental reference in the X axis. The ProtoTRAK RMX CNC allows you to program using either.

5.5   Referenced and Non-Referenced Data

Data is always loaded into the ProtoTRAK RMX CNC 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. A.G.E. Profile Programming requires the Advanced Features with Verify Option.

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 RMX CNC 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.

i00176
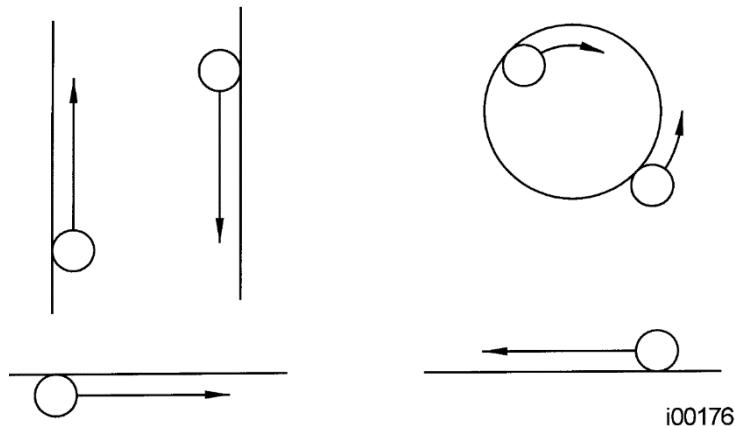
Figure 5.7a - Examples of Tool Right

i00175
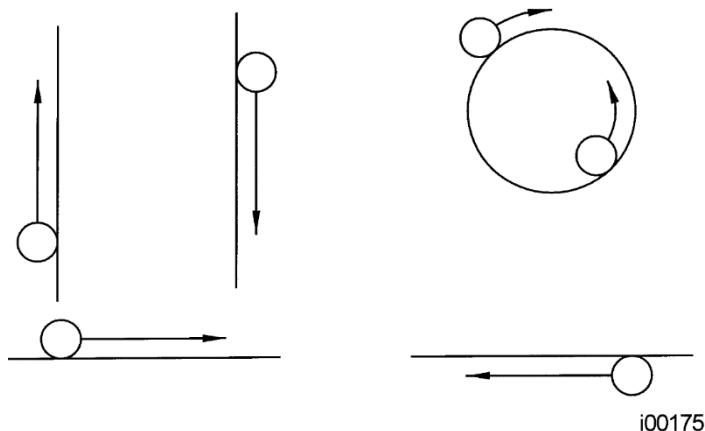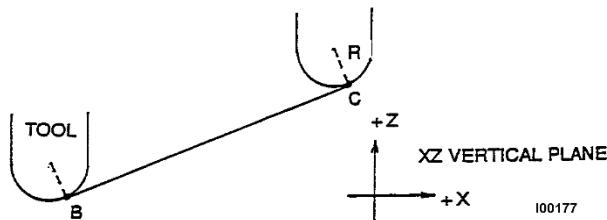
Figure 5.7b - Examples of Tool Left

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 5.8a - 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.
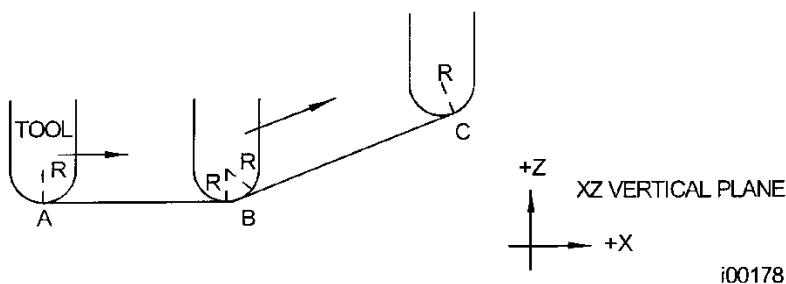
Figure 5.8b - In order to respect the lines defined by the programmed points, the ball End mill never touches point B. Tool starts centered over A offset up by the tool Radius R. It moves right until it is tangent to both AB and BC. Then moves to point C in the first example.

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 5.3).

5.10   Conrad
Conrad is a unique feature of the ProtoTRAK RMX CNC 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.
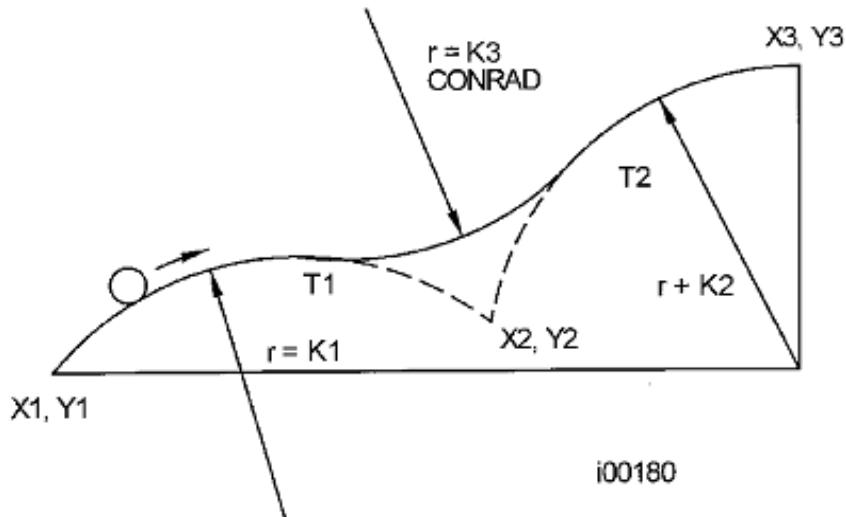
Figure 5.10a - A Conrad is added between the two intersecting lines.

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 RMX CNC 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.
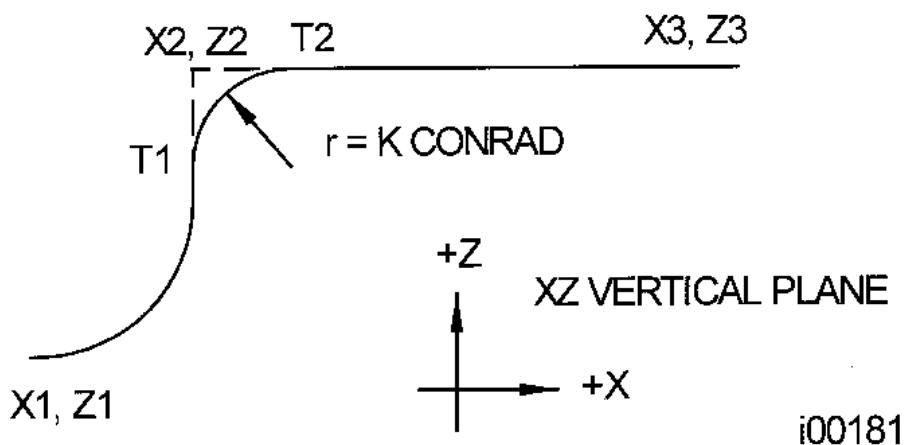
Figure 5.10b - A Conrad is added between an arc and a line.

5.11 Memory and 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 RMX CNC holds any part program that is ready to run. While a program is being written, it is in current memory.

Storage of programs can be done on a USB device or a Network drive. This means a user can store programs to an offline computer that is networked to the RMX CNC. To add a network drive, the user must setup the operating system to do so. See section 14. We strongly recommend you habitually back up programs.

6.0   Info Keys

Info keys on the left side of the screen enable you to:
- Configure your ProtoTRAK RMX to your preferences
- Make changes to system defaults
- Look up information
- Access advanced capability

Info Keys are always available while doing any operation on the ProtoTRAK, you may tap an Info key.

Fly Out Windows

When you tap an Info key, the key will turn yellow and the associated fly out window will appear over the window you had been using. When you are finished with the window, tap the key again to close it.
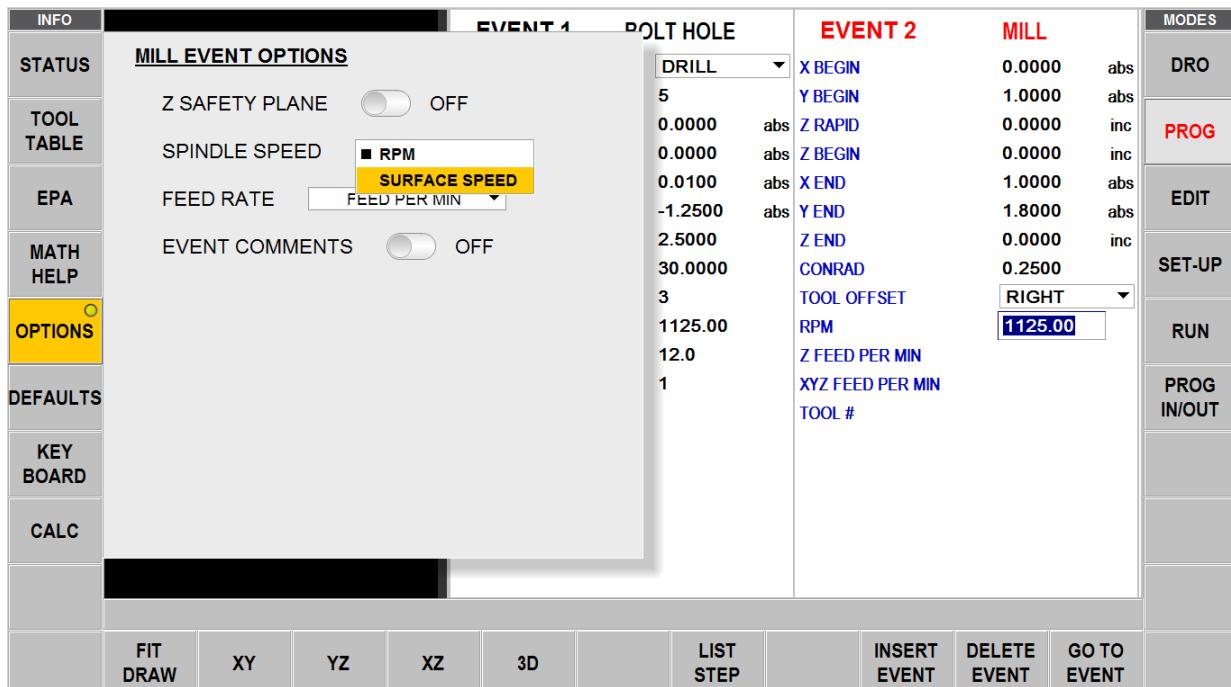
Figure 6.0a - The Option key opens a Fly Out Window over the event programming to give you instant access to relevant choices.

Once you open a Fly Out Window, you must close it by tapping the associated Info Key.

						BASE	SET		
LIB #	TOOL #	ATC LOC #	TOOL TYPE	MATERIAL	# FLUTES	DIAMETER	Z OFFSET	DIA MOD	Z MOD
			PROGRAM TOOLS (1 to 99)						
	1		Face Mill ▼	INSERT ▼	8	3.0000	0.0000	0.0000	0.0000
	2		Finish End Mill ▼	CARBIDE ▼	3	0.7500	0.0000	0.0000	0.0000
	3		Finish End Mill ▼	CARBIDE ▼	3	0.3750	0.0000	0.0000	0.0000
	4		Finish End Mill ▼	CARBIDE ▼	3	0.2500	0.0000	0.0000	0.0000
	5		Countersink ▼	CARBIDE ▼	2	0.5000	0.0000	0.0000	0.0000
			ATC TOOLS						
	▮	1	Face Mill ▼	INSERT ▼	8	3.0000	0.5500	0.0000	0.0000
		2	Finish End Mill ▼	CARBIDE ▼	3	0.7500	0.2500	0.0000	0.0000
		3	Finish End Mill ▼	CARBIDE ▼	3	0.3750	-0.0000	0.0000	0.0000
		9	Tap ▼	HSS ▼	2	0.2500	0.0000	0.0000	0.0000
			TOOL LIBRARY (101 to 199)						
101			Center Drill ▼	HSS ▼	3	0.5000	0.0000	0.0000	0.0000
			Finish End Mill ▼	CARBIDE ▼	2	0.2500	0.7500	0.0000	0.0000

ATC LOC 1   TOOL # 101   DIA 0.5000   Center Drill   TOOL TABLE

INFO · STATUS · TOOL TABLE · EPA · MATH HELP · DEFAULTS · KEY BOARD · CALC

MODES · DRO · PROG · EDIT · SET-UP · RUN · PROG IN/OUT

CALL TOOL · RETURN TOOL · ATC LOC # · CLEAR LIB # · DELETE TOOL · DELETE OFFSET · SHOW ALL ATC · ADD TO LIBRARY · ERASE LIBRARY · JOG

PATH · PART · OFFSET · PICTURES · NOTES · STRAT · CODES

Figure 6.0b - The Tool Table Fly Out makes it quick and easy for you to define your tools as you program.

6.1   Status

The Status Info Key gives you a convenient look at the current status of your ProtoTRAK.

6.1.1   Operational Status

Units – Choose between in. and mm. All dimensional information will appear in the units you choose. If you change units, the change will take effect as soon as you tap the Status key to put the window away.

Axis Mode – From 2 to 3 axis. If you change Axis Mode, the change will take effect as soon as you tap the Status key to put the window away.

Program Name – For reference. You may rename a program in Program In/Out Mode or the Event 0 in the Program Mode.

User Profile – By default, the screen will show Default User as the profile name until individual user profiles are created. Once different user profiles are created, the user profile currently in use is what will be displayed. For more information on User Profiles, see section 6.5.3.

6.1.2   Software Version

This will list both the current software and firmware.

6.1.3   Options Activated

This is a list of all available control options and whether or not they are activated. To purchase an option, call Customer Service at 800-367-3165 or Sales Support at 800-421-6875.

See Section 3.2 for a description of available options.

6.1.4 Features OFF/ON

Advanced Features with Verify – The following features are available when switched "ON" in the Status Fly Out Window or may be turned "OFF" if you wish to simplify the operation of the ProtoTRAK RMX by eliminating features.

- Adaptive Pocket Roughing
- Verify Make Part – solid model graphics of programmed toolpath.
- Finish Tool #
- Rest Machining
- Multiple Fixture Offsets
- Event Comments
- G-code Editor
- Update GCD Program
- Search Edit
- Pictures and notes saved in Part Programs
- Clipboard
- PTG File Types – Save G-code programs along with tool info, pictures, notes
- Copy Events – Repeat, Mirror, Rotate Z-Axis, Drill to Tap Engraving
- Engraving
- Thread Mill

Verify allows you to see a solid model rendition of your programmed part as well as the programmed toolpath. It is very useful to prove out a program and prevent mistakes.

Network – Access to the networked drives may be turned off so that communications between the ProtoTRAK RMX and the network are halted.

6.2 Tool Table

The Tool Table is divided into three sections: Program Tools, ATC Tools, and Tool Library.

Base Tool
This is the status of the Base or reference tool – Set or Not Set. See below for a description of the Base Reference Tool.

Figure 6.2a - *TMC Tool Table - Tool status info located at the top of the window.*

Program Tools

These are the tools that you have entered into the events of your current program. Up to 99 tools may be entered from 1 – 99.

As long as the tool number is in one of your events, it will automatically appear in this part of the screen. See below for how to set up the tool. You may set up the tool at any time while you are preparing the program, including opening the Tool Table window as you are programming while the tool is on your mind. If you forget to enter diameter or other essential information before running, the ProtoTRAK RMX will present you with an error message. To help, the info that is required before run is highlighted in orange if not entered and it turns green when entered.

Tool Offsets and Mods in Program Tools are **not** saved when a current program is closed, erased or if the control is powered off. To save this information, see Save/Open Temp in the Program In/Out Mode for saving a temporary file.

ATC Tools

Used to manage ATC tools from the Tool Table window. Select an ATC tool by clicking on any cell within a specific row, then select CALL TOOL to get the tool from ATC to the spindle.

To return any tool from the spindle back into ATC, select RETURN TOOL.

- **Applicable only on VMC2:**
  ATC Front/ATC Back – This soft key toggles between ATC FRONT and ATC Back. If the ATC is located in the back, then the soft key should display "ATC FRONT." Selecting ATC FRONT, will bring the ATC out forward and vice versa. See figure 6.2b below for reference.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

Figure 6.2b - VMC2 Tool Table

- **Applicable only on TMC's:**

  To index the carousel, select ATC LOC #. A green message reading "Current carousel location is _" will be displayed at the bottom of the window. In the field located under the green message reading "ENTER NEW CAROUSEL LOCATION: _" enter the index value you want your ATC location to face the spindle. See figure 6.2c below for reference.

Figure 6.2c - TMC Tool Table - Example of ATC Tools selected to index the carousel.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

To display a list of all of the indexed ATC Tools' information, select SHOW ALL ATC. This button will also list all of the empty ATC Tools slots.

HIDE EMP ATC button is enabled only after selecting SHOW ALL ATC. Select HIDE EMP ATC to hide information for the empty ATC Tools.

Tool Library

The Tool Library can be a convenient way to save tool information and use the information to cut down on time in doing programming and set up. All the information, including the Offsets and Mods, are saved in the Tool Library. You may set up the tools in the Tool Library separately from a program by entering tool information directly into the Tool Library or you may transfer information from Program Tools as you go. Be sure to fully understand the Tool Library procedures to avoid using tools with incorrect or incompatible offsets so you don't crash a tool!

To input information directly into the Tool Library, simply tap in any of the columns of the empty row and enter information.

To transfer tool information that is already in a Program Tool, simply tap on the Program Tool, Tool # and tap ADD TO LIBRARY.

Tool Library tools may be identified by a number from 101 – 199. We do this to distinguish them from the Program Tools which are numbered from 1 – 99.

Tool Library tools can be sorted by Lib# or Tool Type or tool diameter in ascending or descending order.

Figure 6.2d - Tool Table Displaying the Tool Library on the VMC2

| | INFO | ATC LOC 1 | TOOL # 101 | DIA 0.5000 | Center Drill | | | | MODES |

Tool Table

LIB #	TOOL #	ATC LOC #	TOOL TYPE	MATERIAL	# FLUTES	BASE DIAMETER	SET Z OFFSET	DIA MOD	Z MOD
PROGRAM TOOLS (1 to 99)								
	1		Face Mill ▼	INSERT ▼	8	3.0000	0.0000	0.0000	0.0000
	2		Finish End Mill ▼	CARBIDE ▼	3	0.7500	0.0000	0.0000	0.0000
	3		Finish End Mill ▼	CARBIDE ▼	3	0.3750	0.0000	0.0000	0.0000
	4		Finish End Mill ▼	CARBIDE ▼	3	0.2500	0.0000	0.0000	0.0000
	5		Countersink ▼	CARBIDE ▼	2	0.5000	0.0000	0.0000	0.0000
ATC TOOLS								
		1	Face Mill ▼	INSERT ▼	8	3.0000	0.5500	0.0000	0.0000
		2	Finish End Mill ▼	CARBIDE ▼	3	0.7500	0.2500	0.0000	0.0000
		3	Finish End Mill ▼	CARBIDE ▼	3	0.3750	-0.0000	0.0000	0.0000
		9	Tap ▼	HSS ▼	2	0.2500	0.0000	0.0000	0.0000
TOOL LIBRARY (101 to 199)								
101			Center Drill ▼	HSS ▼	3	0.5000	0.0000	0.0000	0.0000
			Finish End Mill ▼	CARBIDE ▼	2	0.2500	0.7500	0.0000	0.0000

Buttons: CALL TOOL | RETURN TOOL | ATC LOC # | CLEAR LIB # | DELETE TOOL | DELETE OFFSET | SHOW ALL ATC | ADD TO LIBRARY | ERASE LIBRARY | JOG

PATH | PART | OFFSET | PICTURES | NOTES | STRAT | CODES

Side buttons: STATUS, TOOL TABLE, EPA, MATH HELP, DEFAULTS, KEY BOARD, CALC
Right side: DRO, PROG, EDIT, SET-UP, RUN, PROG IN/OUT

Figure 6.2e - Tool Table Displaying the Tool Library on the TMC's

- **Tool Table Status Bar**
  The status bar is located above the tool table heading which displays the following information in each of the four field boxes.

  o **Applicable only on VMC2:**
    Status Bar Box #1: ATC Front/ATC Back – Displays the current position of ATC.
    Status Bar Box #2: TOOL #/LOC # – Information about the tool in spindle. This information will change based on manual tool, library tool, and ATC tool along with ATC LOC. It will also display NO TOOL when there isn't a tool in the spindle.
    Status Bar Box #3: DIA – Lists the diameter of the tool.
    Status Bar Box #4: Lists the tool type.

| | ATC BACK | TOOL # 101 | | DIA 0.5000 | Ctr Drill |
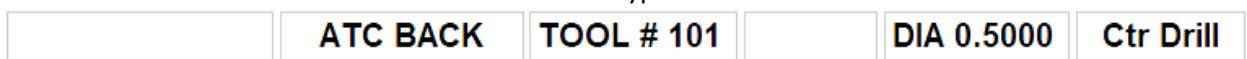
Figure 6.2f - VMC2 – Example of the Status Bar located above Tool Table header

  o **Applicable only on TMC's:**
    Status Bar Box #1: ATC LOC – Location of the ATC relative to the tool change position (position straight across from spindle).
    Status Bar Box #2: TOOL #/LOC # – Information about the tool in spindle. This information will change based on manual tool, library tool, and ATC tool along with ATC LOC. It will also display NO TOOL when there isn't a tool in the spindle.
    Status Bar Box #3: DIA – Lists the diameter of the tool.
    Status Bar Box #4: Lists the tool type.

| ATC LOC 1 | TOOL # 1 LOC 1 | DIA 0.5000 | Center Drill |
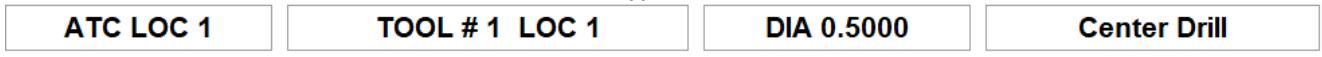
Figure 6.2g - TMC's – Example of the Status Bar located above Tool Table header

- **Tool Table Column Headings**

  LIB# – Used only in the Tool Library section. You assign numbers to the tools in the Tool Library by tapping this field. In order to distinguish Tool Library from tool numbers you assign during programming, the Tool Library tools must be a number from 101 – 199. It is not mandatory to assign a Library Number, but it may help as a cross reference when you assign the Library tool to a Program Tool. If you want to use the library tool in DRO, then a library tool number is required.

  Tool # – The tool number you assigned in the current program, from 1 – 99. These tools are the Program Tools.

  ATC LOC # – Indicates the location of the tool in ATC.

  Tool Type – The type of tool chosen from the drop-down list if you elect to do so.

  Material – Selects the tool material from the drop-down list if you elect to do so.

  # Flutes – Specifies the number of flutes for the cutter. This is elective unless you use Feed per Tooth.

  Diameter – The tool diameter. This is mandatory in order for the ProtoTRAK RMX to calculate the tool path with offsets.

  Z Offset – The difference between this tool and the Base Tool. You must set the z offset to run a program. The field will be orange and there will be a line instead of a number until the tool is set.

  Dia Mod – Allows for small adjustments e.g. for reground tools; this is elective.

  Z Mod – Allows for small adjustments in Z Offset; this is elective.

6.2.1 The Base Tool

For your convenience, we have supplied a 3/8" drill shank with a rounded tip. If you insert the shank into a tool holder and never remove it, this can serve as your Base Tool. Referencing all tool offsets from this Base Tool gives you the ability to use the same tool in several programs without resetting offsets. We also suggest that you establish a point you will use to touch off every time. This simple habit will make tool setting quick, easy and secure.

Each time the ProtoTRAK RMX is powered down, the base tool reference will be lost.

6.2.2 Setting Z Offsets

There are two different methods for setting your Z offsets:

Method 1:

Setting Z Offsets can simply be a matter of first setting the Base Tool and then each tool. You can tell if a tool is not set because the box will be orange and there will be dash instead of a number.

To Set the Base Tool:

1. Put your Base Tool in the spindle.
2. Tap the Z Offset box in the Base Tool part of the Tool Table screen.
3. Touch off your Base Tool. We recommend that you use a touch-off point that you will be able to use for all tools, now and in the future. If you mix up touch off points, you will crash a tool.
4. Press Enter.

Set the other tools Program or Tool Library:
1. Put the tool in the spindle.
2. Tap the Z Offset box for the tool.
3. Touch off the tool to the same place that you touched off the Base Tool.
4. Press SET
5. Repeat for all tools that you want to set or reset

Note - *Just because the offset field is green and there is a number instead of a dash, this is NOT a guarantee that the offset is correct. If you want to start over to be sure, tap the value and tap the DELETE OFFSET key and redo the offset with the procedure above.*

Method 2: (Added in 2.0.0 software version)
You can use our optional Tool Measurement Device to measure your tool offsets. See section 3.5.2.
1. In the Tool Table screen, do not enter a value for the BASE. It should remain orange and read NOT SET.
2. Put your Base Tool in your tool measurement holder.
3. Set your tool measuring device to 0.
4. Use the measuring device to get each tool's offset relative to the base tool.
5. Manually enter the tool offset into the tool table under Z OFFSET, exactly as it read from the measurement device.

6.2.3　　　Using Library Tools While Programming

The Flyout Windows of the ProtoTRAK RMX makes programming your tools easier than ever. At any time, tap the Tool Table key and the window will open over the mode window you are working in. Having a Tool Library at hand makes it even easier.

To use the Library Tools, simply do the following:
1. While programming, enter a number for your Program Tool from 1 – 99. This will create an entry in the Program Tools area of the Tool Table for each unique tool number. The entry will appear as soon as you finish all the prompts for the event.

2. Tap the TOOL TABLE Info key. The Tool Table will open.

3. In the Tool Library section of the Tool Table, tap the Tool # Field in the row of the Library Tool you want to use.

4. Press Enter. The information for the tool in the Tool Library will copy to the Program Tool. If the Library Tool had a LIB #, that number will appear for reference next to the Program Tool #.

That's it!

Note - *We do not allow you to enter the Library Tool # into the Tool prompt while programming an event. We do this as a precaution. Tool Library Z Offsets are stored in the control, but that doesn't mean they are correct. Programs written at different times, on different ProtoTRAKs or on the Offline Software may have offsets stored that are not correct for the machine and tool on which they will run.*

6.2.4 Using ATC Tools While Programming

To use the ATC Tools, simply do the following:

1. While programming, enter a number for your Program Tool from 1 – 99. This will create an entry in the Program Tools area of the Tool Table for each unique tool number. The entry will appear as soon as you finish all the prompts for the event.

2. Tap the TOOL TABLE Info key to open the Tool Table.

3. In the ATC Tools section of the Tool Table, tap the TOOL # Field within the row of the Library Tool you want to use and enter the Tool #

4. Press Enter. The information for the tool in the ATC Tools will copy to the Program Tool. The ATC Tool has an ATC LOC #, that number will appear for reference next to the Program Tool #.

Note - As a precaution, the program does not allow the user to enter the ATC LOC # into the Tool prompt while programming an event. ATC Tools and related information are stored in the control, but that doesn't mean they are correct. Programs written at different times, on different ProtoTRAKs or on the Offline Software may have offsets stored that are not correct for the machine and tool on which they will run.

To prevent errors, we make it a deliberate action for you to relate the ATC Tool to the Program Tool. This will give you the chance to decide whether you want to trust the values or not. It is an extra step, albeit a small one, but well worth it to avoid what could be an expensive crash.

6.2.5 Using ATC Tools and Library Tools While Programming

To use ATC Tools which are also Library Tools do the following:

1. While programming, enter a number for your Program Tool from 1 – 99. This will create an entry in the Program Tools area of the Tool Table for each unique tool number. The entry will appear as soon as you finish all the prompts for the event.

2. Tap the TOOL TABLE Info key to open the Tool Table.

3. In the Library tools section enter ATC LOC # 1-8 *(VMC2)* **or** ATC LOC # 1-16 *(TMC's)*. Whichever is empty, the software will ask you to load the tool into the spindle so it can put the tool in ATC. Once done the ATC Tool row would be created and all the information from the Library tool will be copied over. The LIB # will also be populated. Now the library tool is being used as ATC tool.

4. In the ATC Tools section enter the Program Tool number that will use the ATC tool in Tool # field. All the information of the tool along with LIB # and ATC LOC # will be copied over to the program tool. Now the program tool is an ATC tool and a Library tool. To show that all these tools are in sync if you select one of them, all 3 will be highlighted in a red box.

Note - You CLEAR LIB # or CLEAR ATC LOC # to disassociate program tool from Library Tool and ATC tool respectively. Once you clear ATC LOC # you can use the tool in Tool Room Ops as a manual tool.

	ATC LOC 1		TOOL # 101		DIA 0.5000		Center Drill			

STATUS				TOOL TABLE								DRO
							BASE	SET				
TOOL TABLE	LIB #	TOOL #	ATC LOC #	TOOL TYPE	MATERIAL	# FLUTES	DIAMETER	Z OFFSET	DIA MOD	Z MOD		PROG
	▲			PROGRAM TOOLS (1 to 99)								
EPA		1		Face Mill ▼	INSERT ▼	8	3.0000	0.0000	0.0000	0.0000		EDIT
		2		Finish End Mill ▼	CARBIDE ▼	3	0.7500	0.0000	0.0000	0.0000		
MATH HELP		3		Finish End Mill ▼	CARBIDE ▼	3	0.3750	0.0000	0.0000	0.0000		ET-UP
		4		Finish End Mill ▼	CARBIDE ▼	3	0.2500	0.0000	0.0000	0.0000		
		5		Countersink ▼	CARBIDE ▼	2	0.5000	0.0000	0.0000	0.0000		RUN
	▲			ATC TOOLS								
		▮	1	Face Mill ▼	INSERT ▼	8	3.0000	0.5500	0.0000	0.0000		PROG
DEFAULTS			2	Finish End Mill ▼	CARBIDE ▼	3	0.7500	0.2500	0.0000	0.0000		N/OUT
			3	Finish End Mill ▼	CARBIDE ▼	3	0.3750	-0.0000	0.0000	0.0000		
KEY BOARD			9	Tap ▼	HSS ▼	2	0.2500	0.0000	0.0000	0.0000		
	▲			TOOL LIBRARY (101 to 199)								
CALC	101			Center Drill ▼	HSS ▼	3	0.5000	0.0000	0.0000	0.0000		
				Finish End Mill ▼	CARBIDE ▼	2	0.2500	0.7500	0.0000	0.0000		

CALL TOOL	RETURN TOOL	ATC LOC #	CLEAR LIB #	DELETE TOOL	DELETE OFFSET	SHOW ALL ATC	ADD TO LIBRARY	ERASE LIBRARY	JOG	
PATH	PART	OFFSET	PICTURES	NOTES	STRAT			CODES		

Figure 6.2.5 - Flyout Window - Using ATC Tools & Library Tools While Programming on the TMC's

6.2.6   Soft Keys in the Tool Table

CALL TOOL: To load manual or ATC tool into the spindle, tap CALL TOOL.

RETURN TOOL: To place a tool back into the ATC, tap RETURN TOOL.

ATC LOC #: (*Note - Not applicable on VMC2)* To index the carousel press ATC LOC #.

ATC FRONT/ATC BACK: (*Note - Applicable only on VMC2)* This soft key toggles between ATC FRONT and ATC Back. If the ATC is located in the back, then the soft key should display "ATC FRONT." Selecting ATC FRONT, will bring the ATC out forward and vice versa.

CLEAR LIB #: To erase the Library Tool # reference from the Program Tool. First tap the Program Tool #, and then CLEAR LIB #.

DELETE TOOL: Deletes data for a Library Tool. Tap anywhere on the row first, then tap DELETE TOOL. This soft key toggles between DELETE TOOL and CLEAR ATC #.

CLEAR ATC #: Appears after selecting any program tool. Clears the ATC LOC # for the selected program tool. This soft key toggles between CLEAR ATC # and DELETE TOOL.

DELETE OFFSET: Deletes the offset for the tool. Tap anywhere on the row first, then tap DELETE OFFSET.

SHOW ALL ATC: Displays all of the 16 ATC tools' information, including the empty rows. This soft key toggles between SHOW ALL ATC and HIDE ATC.

HIDE ATC: Hides rows that are empty in the ATC Tools section. This soft key toggles between HIDE ATC and SHOW ALL ATC.

ADD TO LIBRARY: Copies the Program Tool information to the Tool Library.

ERASE LIBRARY: To erase all the Library Tool information at once.

JOG: Turns on the servo motors so you can jog the ram to position for doing the Z offset or X & Y.

Note - The jog feature always defaults to the positive direction.

Note - A flashing orange SAFETY MSG ON soft key will appear when the machine detects any safety alerts. To get rid of the message, simply tap on the SAFETY MSG ON soft key.

6.3   EPA

The EPA (Enhanced ProtoTRAK Assistance) key will give you quick access to a collection of short tutorials that provide help for using the ProtoTRAK RMX CNC. While these tutorials are useful for resolving questions or jogging the memory, they are not intended to substitute for the user manual or for product training.
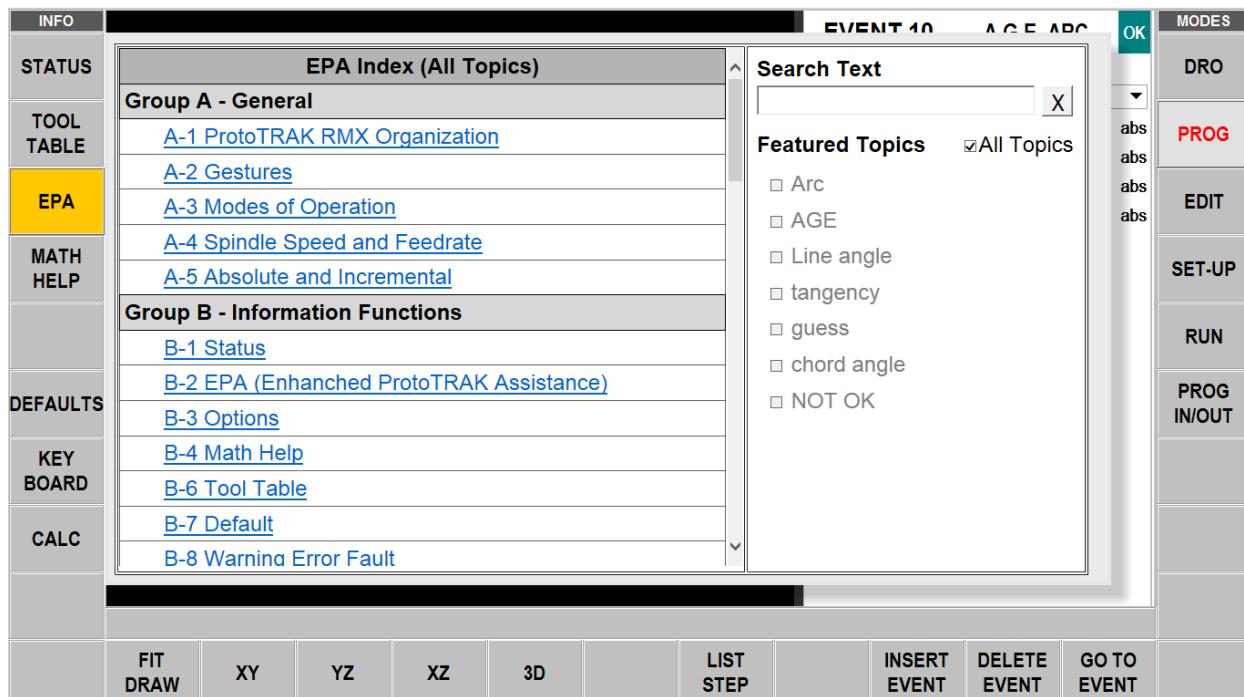
Figure 6.3 - The EPA Screen

6.3.1   EPA Index

The EPA Index will display the tutorials available depending on either the current operation or your selection in the Search Text box.

For example, if you are in the Program Mode, when you press the EPA key, the EPA Index will display the topics most likely to be of interest for Program Mode operations. For example, say you are doing a part program using the AGE (Auto Geometry Engine™) and you forget the meaning of the Line Angle prompt. By pressing the EPA Info key, the EPA Index will appear. Instead of listing all the topics, a shortened list of only those topics likely to be of interest for that operation will appear so you will be able to quickly choose a tutorial with the information you need.

EPA Topics are arranged in Groups:
- A. General – Overall operation of the ProtoTRAK RMX
- B. Information Function
- C. DRO Mode
- D. Program Events
- E. Other Program Features
- F. Edit Mode
- G. Set-up Mode
- H. Run Mode
- I. Program In/Out

6.3.2   Searching the EPA Index

There are two ways to search for topics:

1. Enter a key word into the Search Text box (See section 6.7 Keyboard). Only the applicable tutorials will appear in the EPA Index box. If you don't like the results, try a different word. To clear previously entered text, tap the X to the right of the Search Text box.

2. Select a Featured Topic from the list by tapping the topic word. A check mark will appear to indicate that the selection is successful. Selecting All Topics will display the full list of topics arranged by groups. To deselect, box again.

6.3.3   Navigating Through a Topic
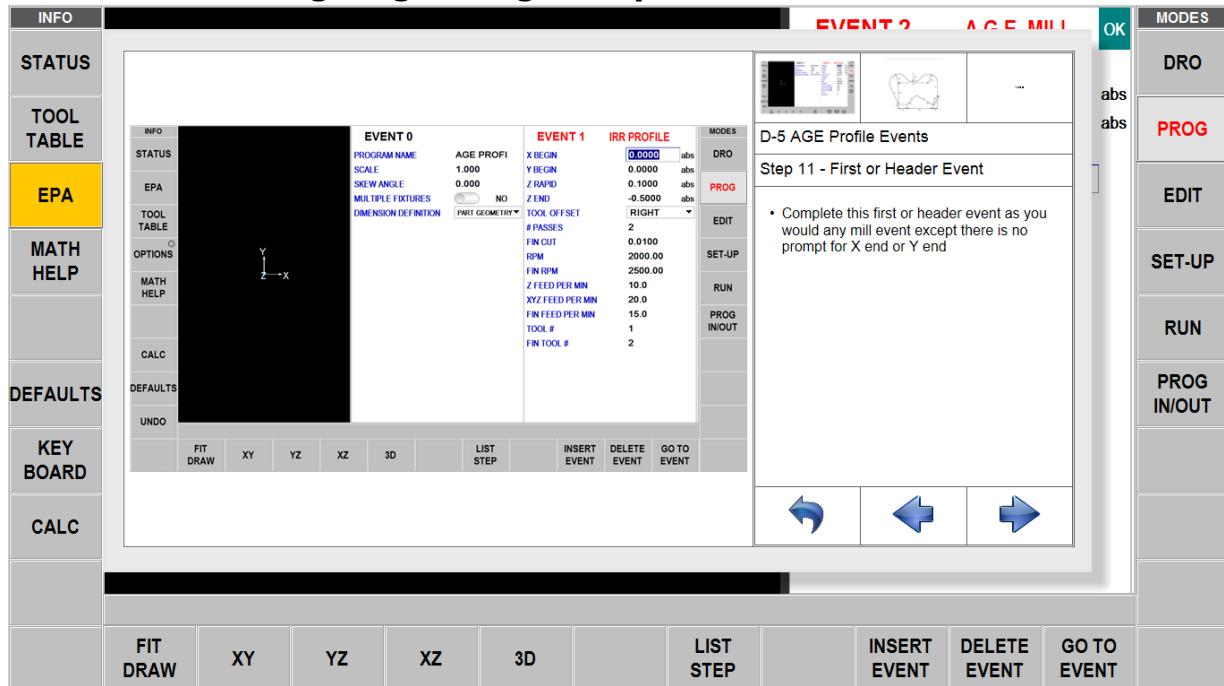
Figure 6.3.3 - An EPA Topics Screen

Display Area: The EPA will display helpful graphics, screen shots or videos as required to help explain the topic. It will be blank if the text does not need illustration.

Items Available: Sometimes there are more than one display available for the topic. Tap the one you want to display.

Use the arrow keys to move through the instructions or to return to the index.

6.4   Math Help
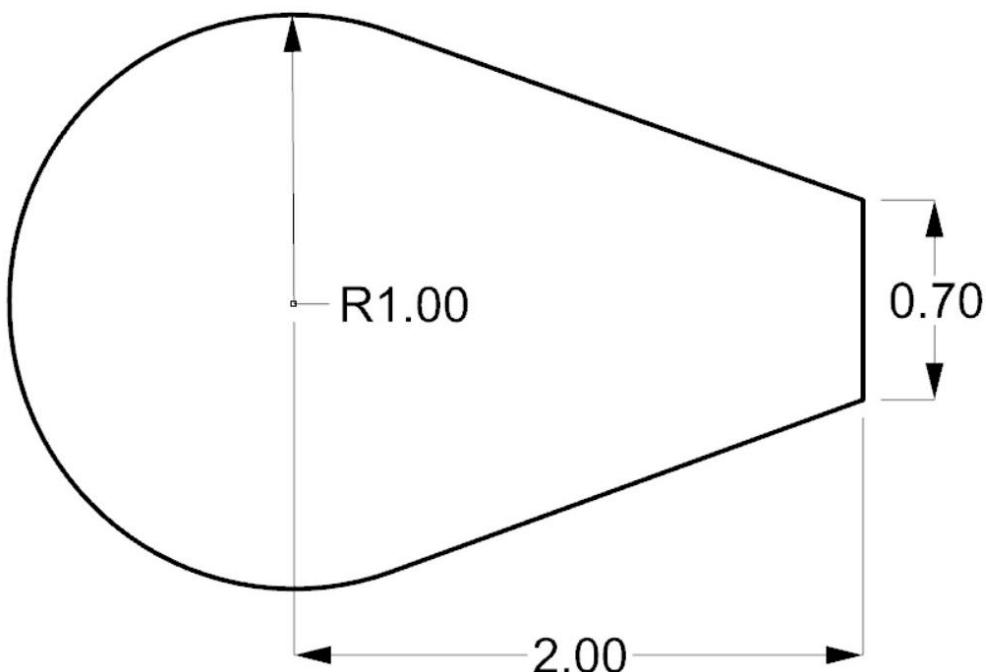
Math Helps are powerful routines that enable you to use data that you have available in order to calculate missing X and Y beginning, ends and centers. The powerful Auto Geometry Engine™ feature will automatically calculate missing print data in most cases, Math Helps provide a powerful companion to that feature.

You may enter into Math Help at any time by pressing the Math Help Info key.
The calculated data may be automatically loaded into your program by entering Math Help while you are programming a beginning, end or center.
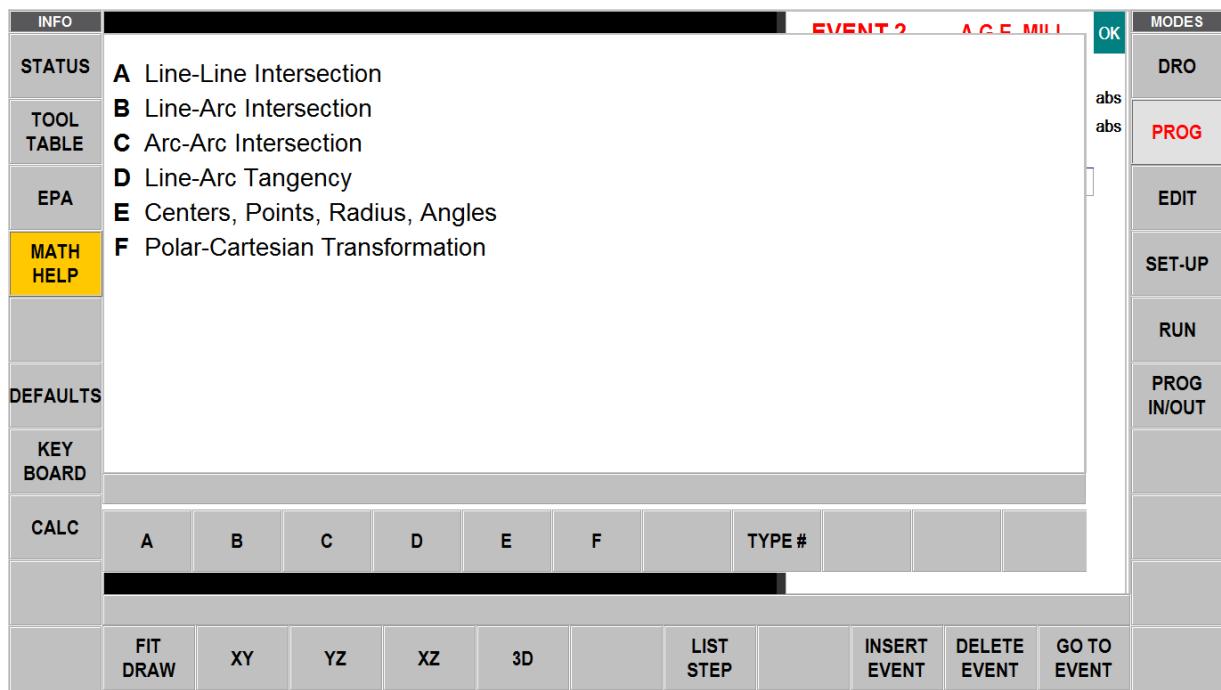
6.4.1 Example – Using Math Help 17

By studying the print in the drawing below, we observe that the drawing lacks the call out for the tangent point between the line and the arc.
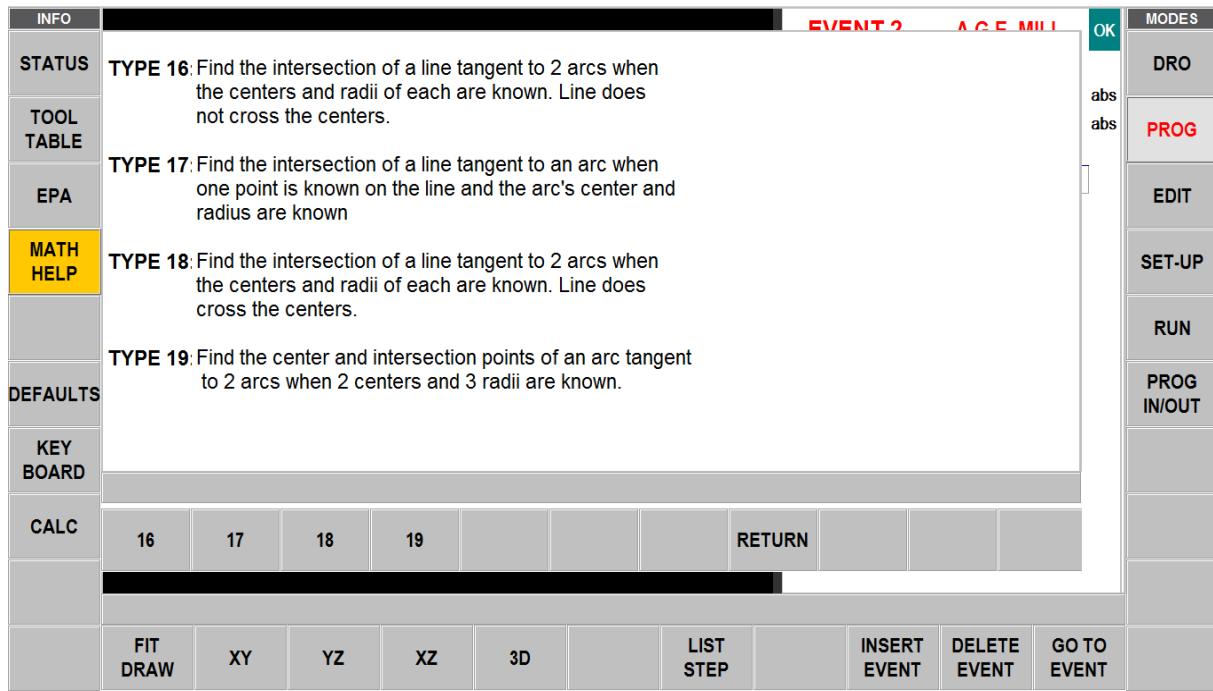
The first screen you will encounter after pressing the Math Help info key will display a list of groups of Math Helps, as shown below.

If you already know which particular Math Help you want to select, tap the TYPE # key and enter the number.

In the example illustration used above, we want to know a point tangent to the line and arc. Therefore, we select Group D Line-Arc Tangency by tapping the D key. The following choices are then displayed:

INFO		EVENT 2 A.C.E. MILL	OK	MODES

TYPE 16: Find the intersection of a line tangent to 2 arcs when the centers and radii of each are known. Line does not cross the centers.

TYPE 17: Find the intersection of a line tangent to an arc when one point is known on the line and the arc's center and radius are known

TYPE 18: Find the intersection of a line tangent to 2 arcs when the centers and radii of each are known. Line does cross the centers.

TYPE 19: Find the center and intersection points of an arc tangent to 2 arcs when 2 centers and 3 radii are known.

Soft keys: STATUS, TOOL TABLE, EPA, MATH HELP, DEFAULTS, KEY BOARD, CALC

Bottom keys: 16, 17, 18, 19, RETURN

Modes: DRO, PROG, EDIT, SET-UP, RUN, PROG IN/OUT

Lower keys: FIT DRAW, XY, YZ, XZ, 3D, LIST STEP, INSERT EVENT, DELETE EVENT, GO TO EVENT

Further study of the print reveals that we have information most suitable for Math Help 17. We select 17 by tapping the 17 soft key.

Enter the known print data as prompted on the data input line. Unlike the event programming, the data must be entered on the data input line near the bottom of the screen. You cannot type the data directly into the list of data as you can in other places in the ProtoTRAK RMX. Use the DATA UP and DATA DOWN keys to move through the list of data.

TYPE 17:

X1	2.0000
Y1	-0.3500
X2	0.0000
Y2	0.0000
R1	1.0000

SOLUTION 1 OF 2

X	0.3351
Y	-0.9422

| LOAD BEGIN | LOAD END | LOAD CENTER | NEXT SOLUTION | EDIT | RETURN | | |
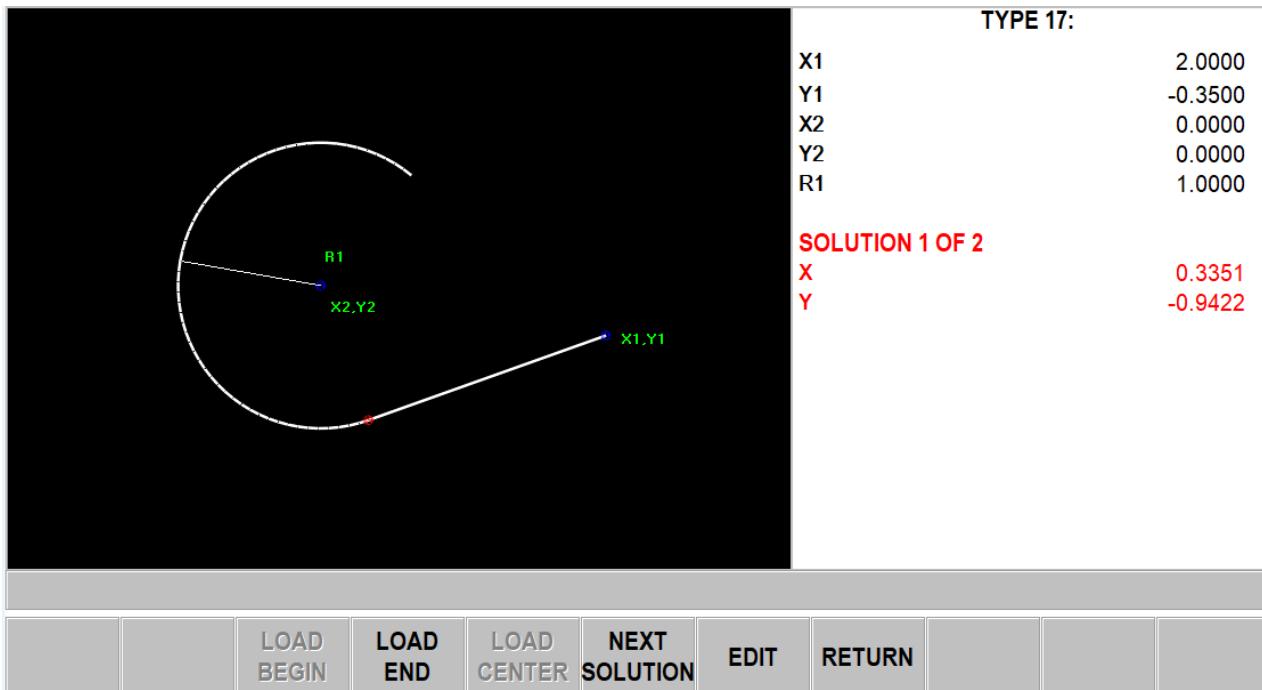
Figure 6.4.1 - Displays Math Type 17 gives us two possible solutions.

There are two theoretical solutions to this problem. The ProtoTRAK RMX will display them in turn. Tap NEXT SOLUTION to look at the alternatives. Looking at the print, you can easily see that we want Solution 1.

If the drawing reveals that you have made a mistake in the data, you may edit your inputs on this screen by tapping Edit. After you correct or change the entry, tap RE-SOLVE.

If you wish to input these values into the program, first tap the number you want and then tap the Load Begin, Load End or Load Center. If you are not in the Program Mode, the Load keys will be greyed out.

6.5   Defaults

Defaults make programming easier by customizing the ProtoTRAK RMX to your machining style and the type of work you are doing. For example, if you usually work in aluminum, you can set your Default Step-over percentage to a larger number so that pockets and face milling events will take less time to run. If you usually work in tool steel, you would set a Step-over percentage to the much lower number appropriate for that work.
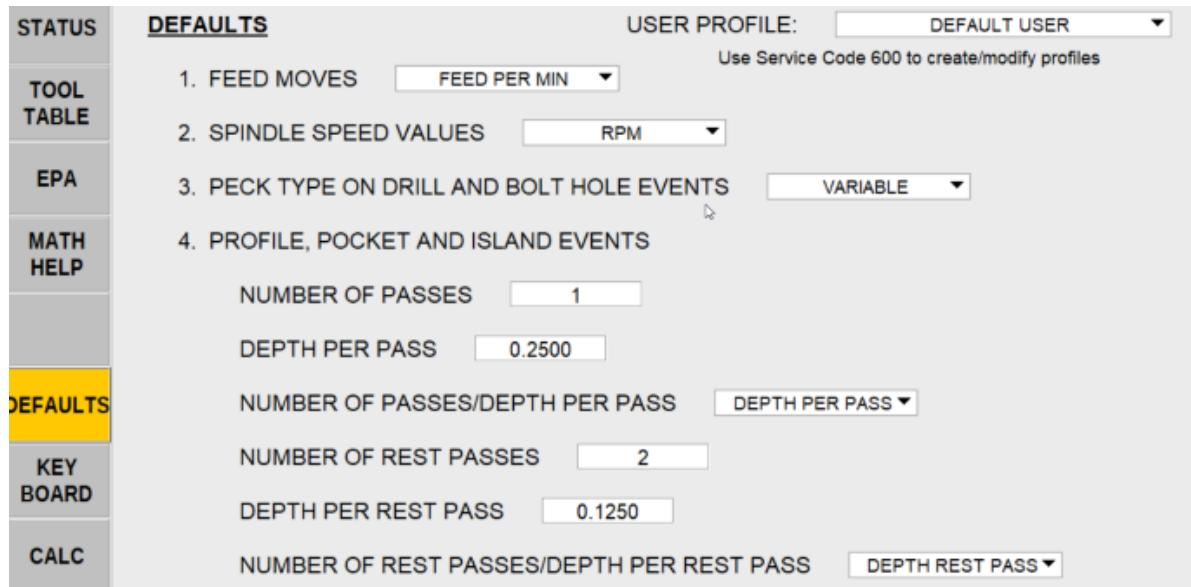
Figure 6.5 - Defaults Flyout Window with the User Profile info displayed on the top corner of the screen

6.5.1 Default Entries and What They Mean

The following are the Default values that you may enter. The number corresponds to the number on the Defaults Flyout Window.

1. **Feed Moves**

   Feed per Min – The Feedrate in Inch per minute or mm per minute.

   Feed per Tooth – The Feedrate in inch or mm per tooth. The number of flutes must be defined for the tool in the Tool Table.

Math Help Type 30 will calculate IPM from the RPM, inch/tooth and number of teeth values you enter.

2. **Spindle Speed Values**

   RPM – Spindle speed in Revolutions per Minute

   SFM or SMM – Surface Speed in Surface Feet per Minute or Surface Meters per Minute, which is the speed of the outside Diameter of the tool.

Math Help Type 31 will convert an RPM value to SFM or SMM.

3. **Peck Type on Drill and Bolt Hole Events**

   Variable – Progressively less each peck with full retract out of hole on each peck

   Fixed – Same depth each peck with full retract out of hole on each peck

   Chip break – Retracts 0.020" between pecks. The pecks are the same as Fixed.

4. **Profile, Pocket, Island and Clear Off Events**

   Number of Passes – Total # of roughing passes to be cut along the Z depth, not including finish cut.

   Depth per Pass – The max depth to be cut for each roughing pass.

   Number of Passes/Depth per Pass – Toggle whether to prompt for # of passes or depth per pass for roughing cuts.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

Number of Rest Passes – The total # of rest (residual) passes to be cut along the Z depth.

Depth per Rest Pass – The max depth to be cut for each rest (residual) pass along the Z depth.

Number of Rest Passes/Depth per Rest Pass – Toggle whether to prompt for # of passes or depth per pass for rest (residual) cuts.

Finish cut in X / Y – The amount of wall material for the finish cut.

5. **Step Over Percentage for Pocket, Island and Clear Off Events**

Step Over Percentage – The percentage of the tool diameter overlap as successive material is removed.

Step Over Percentage – Offset and parallel tool path – This step over percentage should balance the speed of the cut, working the tool properly, and the fact that the cutter will be loaded up more on the initial cut where it will be taking a full width of cut, and when it is in a corner.

Step Over Percentage – Offset & Parallel Tool Path – Z Finish Cut – This helps user define a separate step over percentage for Z floor using offset & parallel tool path.

Step Over Percentage – Adaptive tool path. Adaptive tool path is a complex software routine to achieve all cutting at a constant chip load. This enables you to determine the chip load by the combination of other settings and the ProtoTRAK RMX will automatically calculate the tool path needed to obtain the finished part using that chip load.

Step Over Percentage – Adaptive Tool Path – Z Finish Cut – This helps user define a separate step over percentage for Z floor using adaptive tool path

6. **Face Mill Event**
Step Over Percentage – The percentage of tool diameter overlap as successive material is removed.

Cutting Method:
   o One Way – All machining is in one direction with rapid positioning in between. This enables you to have all climb or conventional cutting for the entire event.
   o Zig Zag – The machining is in a back-and-forth pattern, with no rapid positioning between changes of direction. This has the effect of alternate climb and conventional cutting.

Z Finish Cut – While programming the event, you will only see the prompt, and the default value set here, when the Bottom Finish Cut is set to 'Yes' in Options.

Number of Passes/Depth per Pass – Toggle whether to prompt for # of passes or depth per pass.

Depth per Pass – The depth amount for each pass.

7. **Pocket, Island and Clear Off Events**
All the following Defaults are applied to the powerful Pocket and Island Events.
Cutting Direction:
1 – CW – Clockwise (conventional cutting)
2 – CCW – Counterclockwise (climb cutting)

Z Entry Method:
- o Plunge – Z goes straight down at the Z Feedrate programmed.
- o Helical – the X, Y and Z are automatically programmed to feed into the part in a helical motion.
- o Zig Zag – the Z feeds in a back-and-forth motion.

Tool Path Pattern:
- o Offset – Starts in a logical place depending on the part geometry and takes successive passes based on the Step over percentage.
- o Parallel – Starts in a logical place and will move in parallel lines depending on the part geometry. Moves back and forth in successive passes. In Machining Angle below, you can enter in an angle for the parallel moves if the part feature is not aligned to the X axis.
- o Adaptive (Advanced Features with Verify Option) – Complex software routine that maintains a constant chip load as it is cutting.

Adaptive Tool Path allows you to machine much faster than other tool paths. With other types of tool path, you must limit your feed rate and step over percentages to those cuts where the tool will be loaded the most. A common example of an area with the greatest chip load is the corner of a pocket. That means that the rest of the tool path is cutting at feeds much lower than the cutter could take.

Figure 6.5.1 - (#7) Offset tool path on the left, Adaptive on the right.

Adaptive Machining is powerful software that calculates a tool path to keep the chip load constant. You can set step over percentages and feedrates more aggressively and it is the job of Adaptive Tool Path to move along a toolpath that won't overload the tool.

Cutting Method:
- o One Way – All machining is in one direction with rapid positioning in between. This enables you to have all climb or conventional cutting for the entire event.
- o Zig Zag – The machining is in a back-and-forth pattern, with no rapid positioning between changes of direction. This has the effect of alternate climb and conventional cutting.

Tool Path Pattern – Z Finish Cut: Separate tool path pattern for Z floor

Z Entry Method – Z Finish Cut: Define Z entry method for Z floor

Cutting Method – Z Finish Cut: Define Cutting method for Z Floor

Z Finish Cut – While programming the event, you will only see the prompt, and the default value set here, when the Bottom Finish Cut is set to 'Yes' in Options.

Z Finish Cut Tool – Select if you want to use Finish Tool or Rough Tool.

Helical Entry Pitch – Input the pitch of the Helix used when starting a pocket.

Zig Zag Entry Angle – Input the angle of the back-and-forth motion used in starting a pocket.

Helical Entry Pitch – Z Finish Cut: Define Helical Entry Pitch for Z floor

Zig Zag Entry Angle – Z Finish Cut: Define Zig Zag Entry Angle for Z floor

Machining Angle – This is only needed when Toolpath Pattern (above) is set to Parallel. It is useful when the part or the pocket is not aligned with the X axis. The Machining Angle allows you to have a toolpath that better suits the part.

Machining Angle in XY – Z Finish Cut – Define Machining Angle in XY for Z floor.

Order of Passes – This is the strategy for when the pocket cannot be completed in a complete pass without picking up the tool. For example, a roughing tool is used that cannot fit in the entire pocket in one continuous path. You have these choices:
- o Depth – Completes each depth for all regions before moving to the next depth. In other words, completes each pass for each region.
- o Region – Completes all passes in each region before picking up the tool. In other words, the part is machined to final depth in each region before starting the next region.

Roughing Link for Rapid – This works with the Parallel and Adaptive Toolpath Pattern. When optimizing tool load, the toolpath is not always contiguous, sometimes the tool needs to move from where it is in to a different section. Those moves between sections are called links. As a default, the ProtoTRAK RMX will handle links in this way:
- o For a rectangular pocket it will lift the tool to Z rapid and move at rapid to the next section and rapid down to 0.020" above pass depth and then feed to the final depth.
- o For an irregular pocket, it will lift the tool to Z Rapid (or the Z Safety Plane), then move at rapid to the next section and rapid down to 0.020" above pass depth and then feed to the final depth.

If you do not care that the tool stays into the cut between sections on the rough cut, you would enter a small value into this prompt. If the links are large, the part may be machined more quickly by having a rapid move between sections. The next prompt will allow you to preserve your finish at the bottom of the cut.

Note - *In Figure 6.5.1 (7) above, the link length is set to a small value, hence the many rapid moves.*

Bottom Z Finish link length for Rapid – This default works with Parallel and Adaptive Toolpath Pattern and is applied when the Bottom Z Finish Cut is set to ON in Options. You may want to minimize the tool marks that occur when the tool lifts from the cut and then reenters. To keep the cutter down into the part, enter a value here that is big enough to include your largest Link length.

Minimum Curvature Radius – This value works with the Adaptive Tool Pattern. In order to keep the load constant on the tool, the Adaptive Toolpath often moves along a series of arcs. The Adaptive Toolpath will try to fit into any section where the tool fits. This could result in the toolpath whittling away at a narrow area that would be better left for a finishing tool of a smaller radius. The Minimum Curvature Radius allows you to set a minimum radius of the arcs to preserve more Rest area for a smaller tool.

Minimum Curvature Radius – Z Finish Cut: Define Minimum Curvature Radius for Z floor.

Note - *Rest area is simply the material that is left over from the rough cut, that is, the "rest" of the material. Apparently, someone got a patent on the use of the word "residual".*

Cut Tolerances are the tolerances that are used when calculating a toolpath. Depending on the complexity of the geometry the calculated toolpath that the machine will follow could deviate by these amounts. These are not machine tolerances.

These tolerances are applied to Parallel and Adaptive toolpaths. For Offset toolpaths the typical accuracy of the calculations should be ~1e-6. For most parts such as circles, rectangles and non-complex geometry there should be little to no deviation and the calculated toolpath is extremely accurate. All Semi-Finish and Finish passes should have the same ~1e-6 accuracy when calculating the toolpaths.

ROUGH CUT TOLERANCE – WITH FINISH CUT – This is the amount of deviation that is allowed from the calculated roughing toolpath. The roughing toolpath is the programmed part minus the finish cut amount minus the cut tolerance. A larger tolerance allows the ProtoTRAK RMX control to calculate the tool path faster, but with the possibility of having larger scallops that would be cleaned up automatically with the Semi-Finish pass. Lowering this value will increase the time to calculate the tool path but also may allow a tool to fit into tighter areas and allow the rough tool to remove more material before the REST tool comes in to finish the remaining material. Depending on the shape and complexity of the part, the rough cut dimension could vary from the programmed dimension minus the finish cut amount to the programmed dimension minus the finish cut amount minus 2 times the cut tolerance. You are guaranteed that the rough cut will never enter the finish cut material.

ROUGH CUT TOLERANCE – WITH NO FINISH CUT – The concept is the same as above, but a smaller tolerance should be entered since this value is only applied when you are machining a pocket or island with no finish cut. This ensures the pocket will be machined accurate to within this tolerance.

REST CUT TOLERANCE – Machining REST material is the same as the ROUGH CUT TOLERANCE – WITH FINISH CUT. Since you will use the same tool for removing REST material as for finish cuts, this setting allows you to machine faster until the toolpath transitions to the Finish Cut.

8. **Circle Pocket Event**
Tool Path Patterns:
   o Offset
   o Spiral (Under the Z entry method, Helical will not work for spiral tool path.)

Tool Path Patterns – Z Finish Cut: Define Tool Pattern for Z floor in Circle Pockets.

9. **Engrave Event – XY Reference**
Engrave from XY position set as the center or lower left reference point.

10. **Subroutine and Copy Events**
These defaults are handy for people who use subroutines events in standard ways. For example, using subroutines to repeat programmed events for Finish Cuts. You will be able to override the default value at the subroutine prompts.

Percent Speed – Origin event(s) Spindle Speed times this value.
Percent Feed – Origin event(s) Feedrate times this value.
Mirror direction:
   o Forward – The toolpath will be mirrored and therefore will be the opposite of the toolpath of the origin event(s).
   o Backward – The toolpath will not be mirrored and will maintain the same climb or conventional direction as the origin event(s).

11. **Multiple Fixture Default Number**
The number of the fixture you are placing at this location. This may be handy if you have multiple fixtures for the same part or operation.

12. **Reference Positions**
Limits – Allow you to set default software limits that are relative to the X, Y and Z Absolute values you set. The ProtoTRAK RMX will not go past these limits.
Home position – In Run Mode, the ProtoTRAK RMX will return to these positions. These numbers are relative to the Absolute 0 set in DRO Mode.

13. **Units Inch/mm**
Setting the default will have the ProtoTRAK RMX operate in that unit each time it is turned on.
Note - *You may change your current operations from one unit to the other with the Status Info key.*

14. **Start up in 2 or 3 Axis Mode**
This default determines how the ProtoTRAK RMX starts after being powered down. You can change this in the Status info key.

15. **Maximum Rapid Feedrates**
If you do not want the maximum of 400 inches per minute you can set the maximum here.
Note - *The Z axis runs at 250 ipm.*

Maximum Feedrate Production OPS (in/min) – Default = 1,000 ipm / 25,400 mmpm
Maximum Feedrate Tool Room OPS (in/min) – Default = 400 ipm / 10,160 mmpm

16. Tool Compensation
- o   Center – No tool compensation, the tool will move along the programmed toolpath.
- o   Right – The tool moves to the right of the programmed toolpath, offset by the radius of the tool diameter set in the Tool Table.
- o   Left – The tool moves to the left of the programmed toolpath, offset by the radius of the tool diameter set in the Tool Table

Note - *See Section 5.7 for an illustration example of Tool Diameter Compensation.*

17. Parasolid Set Z Offsets

Used when opening a Parasolid file, and setting Z values relative to your solid, using the VIEW SOLID feature.
- o   Set X Rapid – Default value to use for Z rapid.
- o   Thru Mill Depth – Default offset amount to use for End Mills when choosing WITH OFFSET.
- o   Thru Drill Depth – Default offset amount to use for Drills when choosing WITH OFFSET.

18. Z Rapid

Z Rapid is the Z dimension to transition from rapid to feed. 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. Remember, when using part geometry programming, two milling events are not connective unless they lie in the same plane.

19. Z Safety Plane

Default value for Z safety plane can be set here and for each event you can turn the Z safety plane ON/OFF and it will use the value from defaults.

20. Z Retract / Aux Event
- o   Z Retract – *Applicable only on VMC2*
    Distance from Part Zero – Default value for Z Retract where tools go for tool changes.

- o   Aux Event – *Applicable only on TMC's*
    X Location for Part Change Position – Sets the part change position along the x-axis relative to x home.

21. Show Supported File Types Only

Can be turned ON/OFF. When turned OFF, the user can see all file types in PROG IN/OUT, select the file, and open it as any file type supported by ProtoTRAK.

6.5.2   Working with Defaults
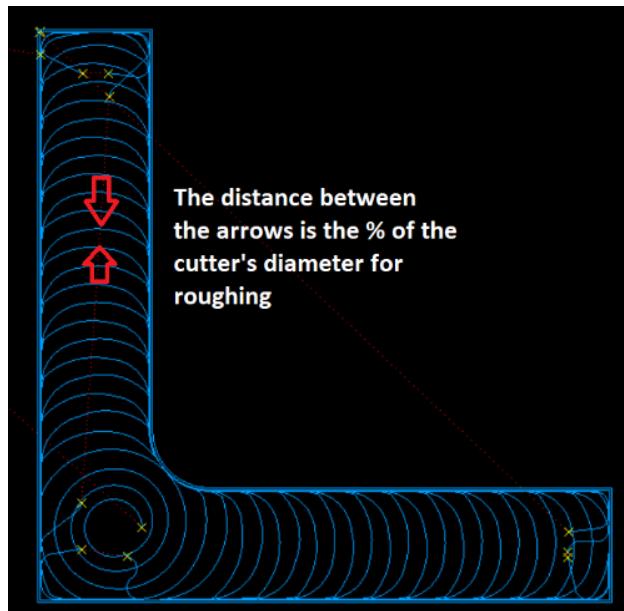
As you are programming, press INC SET or ABS SET to accept the default value, or simply enter another value and press SET.

Whatever you choose as a default can be easily changed in defaults or overridden event by event with the prompt or in the Options Flyout window.

If you change Default value in the middle of the program, the new Default value will take effect on the Event you are on when you make the change. For example, say you have the Default value TOOL PATH PATTERN set to ADAPTIVE from Event 1 to Event 5. While programming Event 6, you change the Default from ADAPTIVE to OFFSET. The Adaptive tool path would apply to Events 1 – 5 and the Offset tool path would apply to Event 6 and thereafter.
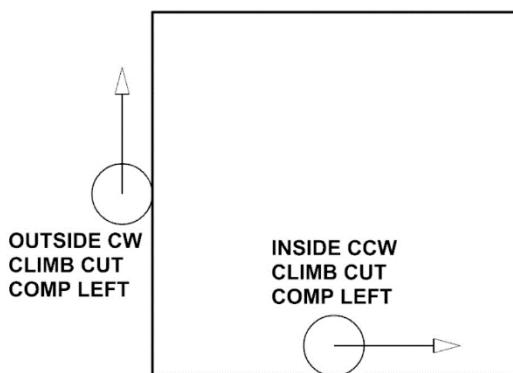
The following are some practical examples of how to use Default values.

1. Step Over Percentage – Adaptive toolpath set to 30% of the cutter diameter

The distance between the arrows is the % of the cutter's diameter for roughing

This illustration is the adaptive roughing strategy for a 0.50" tool at 30% step over. That means the width of cut (step-over) per circular cutting pass is only 0.150" but the depth of cut is 0.50". The feed rate can be 290 IPM @ 5000 RPM, resulting in over 10 cu. Inches per minute of metal removal. 3.8 HP is all that is required.

2. Cutting Direction

OUTSIDE CW
CLIMB CUT
COMP LEFT

INSIDE CCW
CLIMB CUT
COMP LEFT

Pockets:
Set to 1- CW to conventional cut
Set to 2- CCW to climb cut
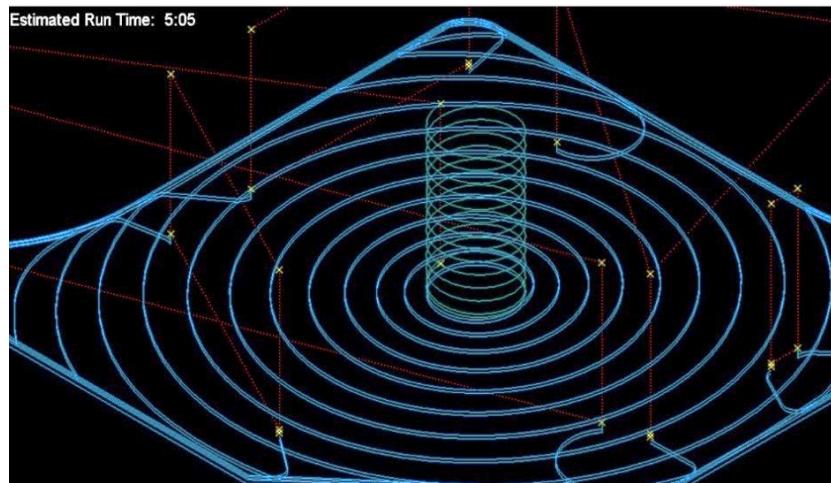
Profiles:
Set to 1- CW to climb cut
Set to 2- CCW to conventional cut
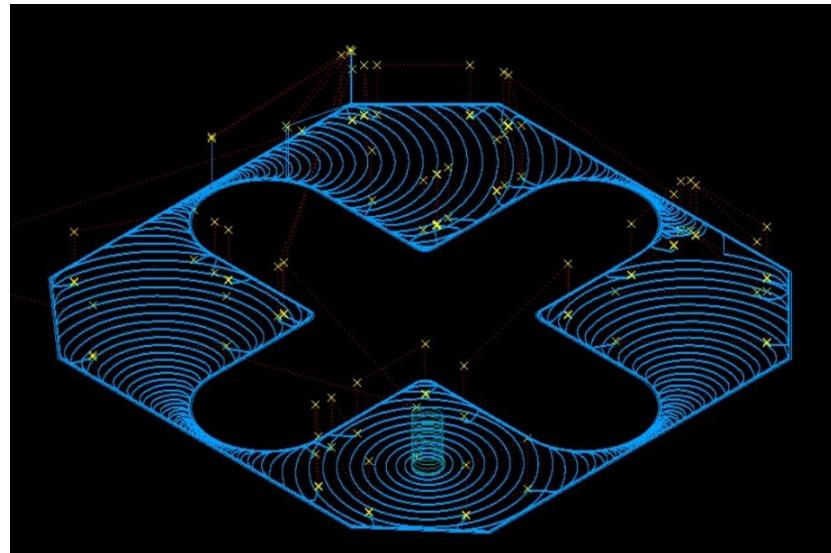
3. Z Entry Method – Helical
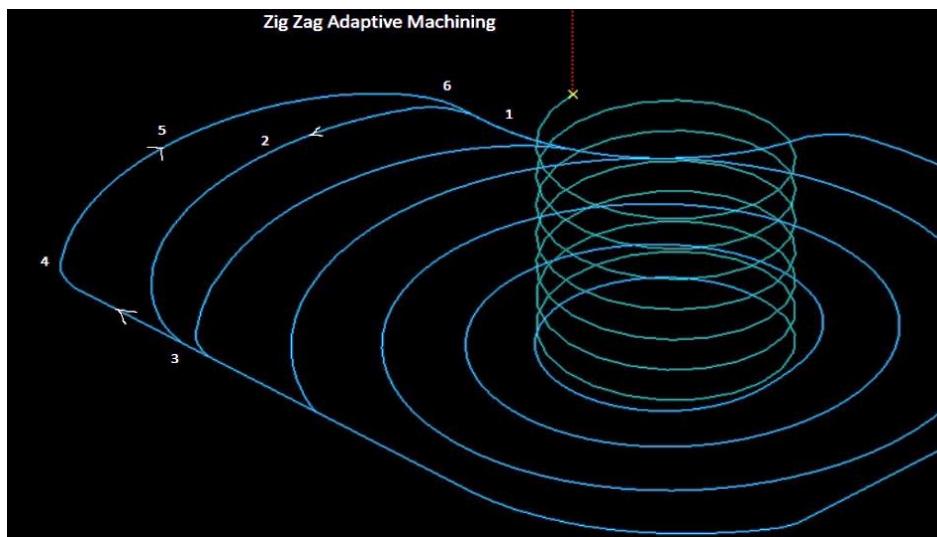
By setting your Z entry type to helical, you can often get to the Z cutting depth faster. For instance, a 0.50" cutter in aluminum using a helix entry, with a pitch of 0.040, can run at 2500 RPM and a feed of 126 IPM. This can be done with the same tool you use to rough with so there is no tool change required.

4. Default Toolpath Pattern – Adaptive

Adaptive roughing is the fastest method of stock removal, with the least amount of wear and tear on both the tool and the machine. Notice the sweeping arc movements. This cutting strategy is extremely efficient.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

5. Default adaptive cutting pattern set to Zig-Zag
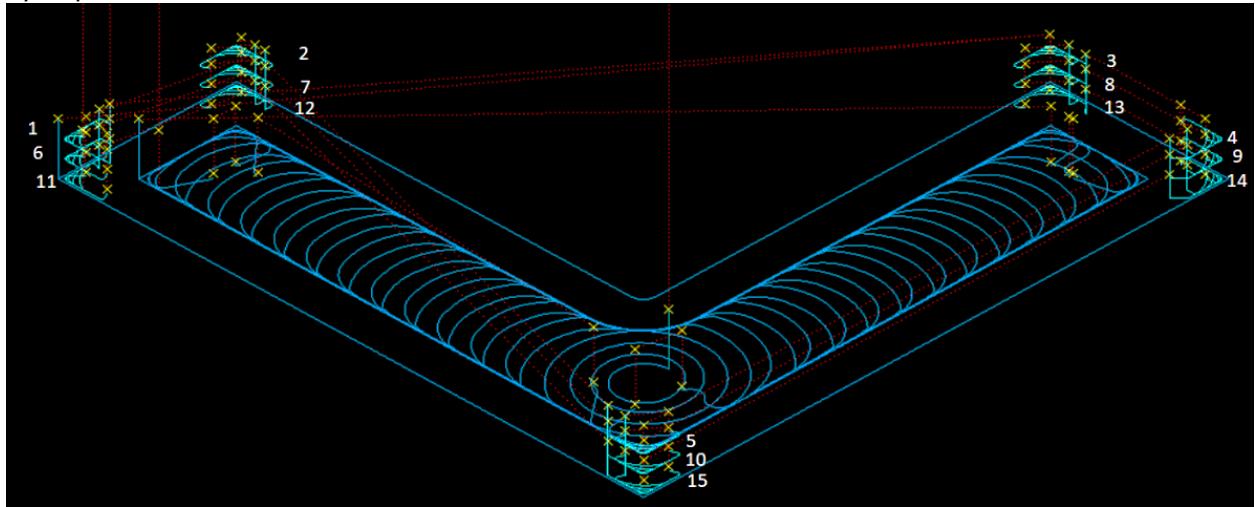
This picture illustrates Adaptive Zig Zag roughing strategy. By combining climb and conventional cutting, the tool will stay down and, in the cut, and require fewer moves to reposition.

6. Helical Pitch Entry
The helical entry pitch has a strong bearing on how fast you can feed your tool down to the required Z depth. The smaller the pitch, the faster the feedrate. We recommend you start with a 0.040 pitch for softer materials and half of that for harder materials.
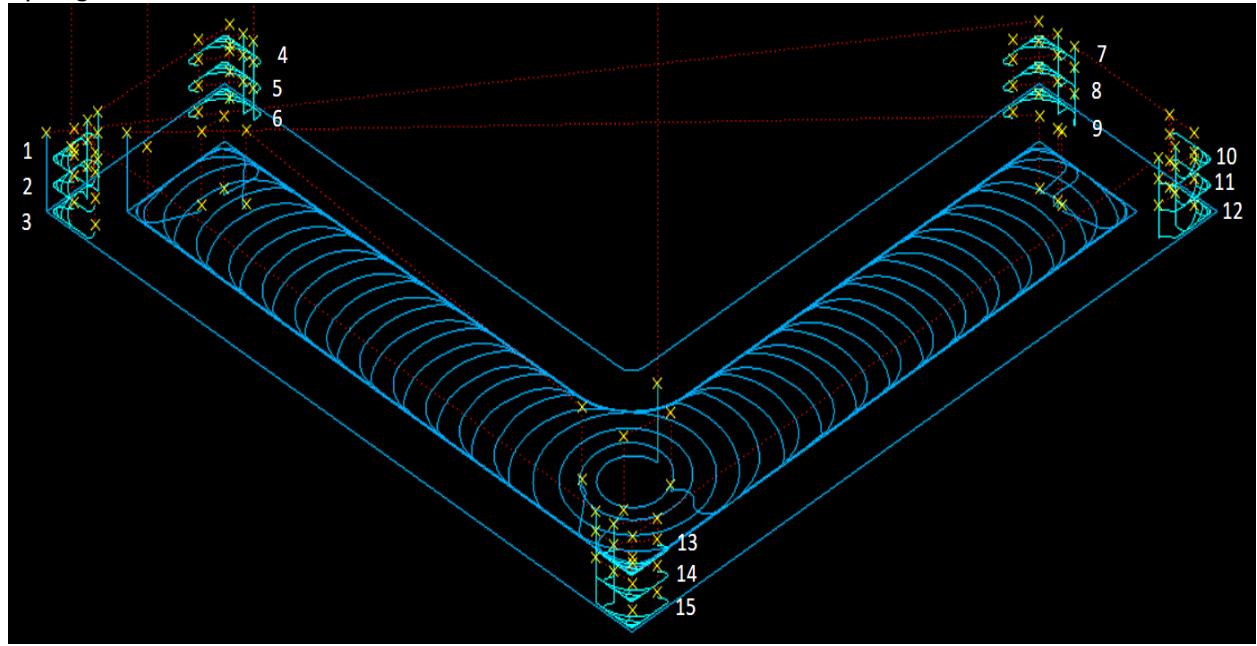
7. Order of Passes

By Depth:
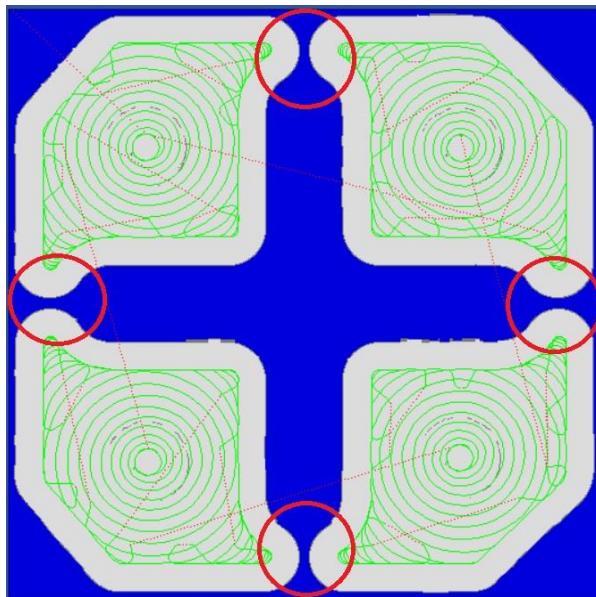
All roughing and rest cuts are made a one Z level before proceeding to the next Z level, as shown in this illustration. Notice how the tool moves to all 5 regions at each Z depth. This makes a total of 14 rapid moves.

By Region:
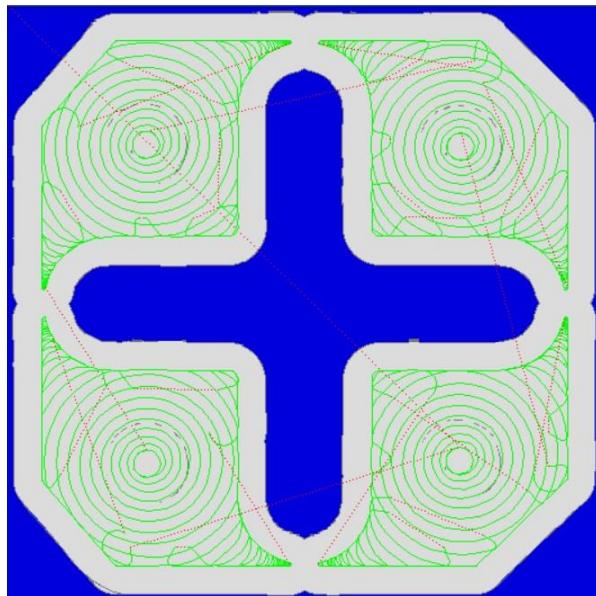
All Z roughing and rest cuts are made in one Z region before proceeding to the next region, as shown in this illustration. This strategy makes a total of only 5 rapid moves. There is minimum of traverse-wasting time. Notice cuts 1,2 and 3 are all in the same corner. The other corners are the same.

8. Minimum Curvature Radius

Above is a part cut with the standard minimum curvature radius of 0.10, utilizing a 0.50" cutter. Notice the areas circled. The cutter could not fit the toolpath between the pocket and the island. This leaves a large amount of stock that the tool will have to cut through at full depth and full width of cut. At high feeds, this may damage the cutter.

Below the minimum curvature radius was changed from 0.10 to 0.010. The toolpath can now fit between the pocket and the island. This slight adjustment allows the use of the larger 0.50" cutter to do the heavy machining you would want to accomplish with the largest cutter diameter possible. Your maximum feed rate will have to decrease but the cycle time will be faster overall.

6.5.3   User Profiles

Each user can set and save their individual default profiles. Service Code 600 allows you to edit and manage User Profiles by using the ADD PROFILE, DELETE PROFILE, and RESET PROFILE buttons or selecting a user profile to change the user name.

Figure 6.5.3 - Service Code 600 – Manage User Profiles Window

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

6.6   Options

The Options Info Key is active when you are in the Program Mode and you have choices for how you program a particular event. If it is not active, the space will be blank. Options give you additional choices for programming events and also allow you to override the default programming style. For example, if you have set the default Drilling to Variable Peck and you wish to change that to Fixed for this particular event, you can quickly do that in Options.
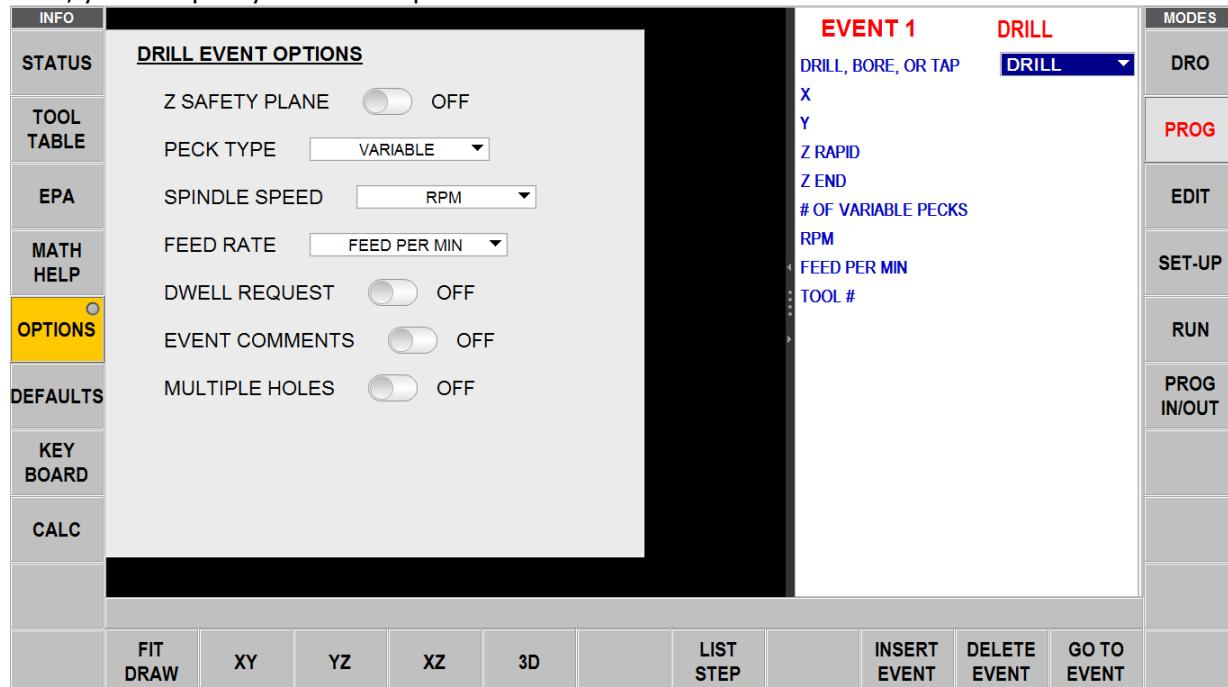
Figure 6.6 - Options enables you to override the defaults on the fly

You set your overall programming style in the Defaults Info Key (Section 6.5 above), but change them on the fly in Options if you wish. If you find yourself frequently changing a default value, we highly recommend that you change the value in the Defaults Info Key.

Note - *Changing a Default does not change a current program. For a changed Default value to appear in a program, the program must be opened or started after the change.*

6.6.1   Programming Variables Found in Options

The options key is active when there are options for what is active in the window you are working in. In programming, the Options that appear when you tap the Options Info key depend on the event type.

Z Safety Plane

The Z Safety Plane is a setting to ensure that every tool will not accidently contact fixtures that stick up above the part. It is the distance from the part's Z Absolute Zero to the bottom of every tool in the spindle. When the Z Safety Plane is set to ON, a prompt will appear for entering the Z Safety plane value.

During program Run, the tool will move from Z Retract to Z Rapid for the first event. Between events, it will move to Z Safety Plane (instead of Z rapid) prior to an X, Y move.

Spindle Speed
- RPM – Spindle Speed for the event in Revolutions per Minute.
- Surface Speed – Spindle Speed for the event in Surface Feet per Minute, which is the speed of the outside Diameter of the tool.

You may use Math Help Type 31 to convert an RPM value to SFM.

Event Comments
Turn this on in order to get the prompt to enter comments you wish to appear during program Run.

Peck Type
Override the default peck for the Drill event:
- Variable – progressively less each peck
- Fixed – same depth each peck
- Chip break – retracts 0020" between pecks

Chip break is the same as fixed but with a 0.020 retract between pecks.

Feed Rate
Overrides the default feed rate setting for the event:
- Feed per minute in inches or mm per minute
- Feed per tooth

Dwell Request
Add a Dwell Time prompt to the Drill and Bore events measured in seconds. The dwell improves the finish on the floor of the hole.

Z Finish Cut Tool
User can choose to use Rough Tool or Finish Tool for Z finish cut.

Bottom Finish Cut
Turn this on to add a Z Finish Cut prompt in the Face Mill, Pocket or Island event. If a Z Finish Cut default was entered in the Defaults it will appear in the prompt.

Cutting Method
Overrides the default cutting method for the event.

Cutting Method:
- One Way – All machining is in one direction with rapid positioning in between. This enables you to have all climb or conventional cutting for the entire event.
- Zig Zag – The machining is in a back-and-forth pattern, with no rapid positioning between changes of direction. This has the effect of alternate climb and conventional cutting.

Step Over Percentage
Overrides the default step over percentage for the event.

- Step Over Percentage – The percentage of the tool diameter overlap as successive material is removed.

- Step Over Percentage – Offset and parallel tool path – This step over percentage should balance the speed of the cut, working the tool properly, and the fact that the cutter will be loaded up more on the initial cut where it will be taking a full width of cut, and when it is in a corner.
- Step Over Percentage – Adaptive tool path. Adaptive tool path is a complex software routine to achieve all cutting at a constant chip load. This enables you to determine the chip load by the combination of other settings and the ProtoTRAK RMX will automatically calculate the tool path needed to obtain the finished part using that chip load.

Machining Angle in XY

Sets the angle the cutter will remove material from a pocket or island. 0º means it will rough along the X axis, 90º degrees means it will rough along the Y axis. You can select any angle between 0 and 90º.

Z Entry Method

Overrides the default Z entry method for the event.

Z Entry Method:
- Plunge – Z goes straight down at the Z Feedrate programmed.
- Helical – The X, Y and Z are automatically programmed to feed into the part in a helical motion.
- Zig Zag – The Z feeds in a back-and-forth motion.

Tool Path Pattern

Overrides the default tool path pattern for the event.

Tool Path Pattern:
- Offset – Starts in a logical place depending on the part geometry and takes successive passes based on the Step over percentage.
- Parallel – Starts in a logical place and will move in parallel lines depending on the part geometry. Moves back and forth in successive passes. In Machining Angle below, you can enter in an angle for the parallel moves if the part feature is not aligned to the X axis.
- Adaptive (Advanced Features with Verify Option) – Complex software routine that maintains a constant chip load as it is cutting.

Tool Path Pattern – Z Finish Cut

User can have different tool path pattern for Z Floor especially for Pockets and Islands.

Order of Passes

Overrides the default order of passes for the event. This is the strategy for when the pocket cannot be completed in a complete pass without picking up the tool. For example, a roughing tool is used that cannot fit in the entire pocket in one continuous path.

You have these choices:
- Depth – Completes each depth for all regions before moving to the next depth. In other words, completes each pass for each region.
- Region – Completes all passes in each region before picking up the tool. In other words, the part is machined to final depth in each region before starting the next region.

Start Location

When using adaptive tool paths, you can turn on Start Location prompts that allow the user to program where the tool enters the workpiece. Turning this option on, adds an X START and Y START prompt within your event.

Multiple Holes
Turn on to have multiple holes during the drilling event.

Number of Passes/Depth Passes
Toggle whether to prompt for # of passes or depth per pass for roughing cuts.

Number of Rest Passes/Depth Per Rest Pass
Toggle whether to prompt for # of passes or depth per pass for rest (residual) cuts.

Rough Cut Tolerance – w/ Finish Cut
This is the amount of deviation that is allowed from the calculated roughing toolpath. The roughing toolpath is the programmed part minus the finish cut amount minus the cut tolerance. A larger tolerance allows the ProtoTRAK RMX control to calculate the tool path faster, but with the possibility of having larger scallops that would be cleaned up automatically with the Semi-Finish pass. Lowering this value will increase the time to calculate the tool path but also may allow a tool to fit into tighter areas and allow the rough tool to remove more material before the REST tool comes in to finish the remaining material. Depending on the shape and complexity of the part, the rough cut dimension could vary from the programmed dimension minus the finish cut amount to the programmed dimension minus the finish cut amount minus 2 times the cut tolerance. You are guaranteed that the rough cut will never enter the finish cut material.

Rough Cut Tolerance – No Finish Cut
The concept is the same as above, but a smaller tolerance should be entered since this value is only applied when you are machining a pocket or island with no finish cut. This ensures the pocket will be machined accurate to within this tolerance.

Rest Cut Tolerance
Same as above, except this allows you to choose a separate tolerance solely for rest mill operations.

Mirror
Turn on to set mirror axis and engrave a mirror image.

Angle
Turn on to engrave at a set specific angle.

XY Reference
Set XY reference point position at which the engraving is done.

Orientation
Set orientation of engraving text as horizontal, vertical, or radial.

6.7 Keyboard

Tapping the KEYBOARD Info key will bring up the alphanumeric keyboard. Rather than being a Fly out window, the keyboard is a separate window that can be dragged around the screen for convenience. After the keyboard is opened, tap the field in which you want to type text or numbers. CAPS LOCK button key option is available. What you type must be set into memory using the Enter key (which is taken as an Abs Set entry). You may use the Abs Set and Inc Set keys on the keyboard as well as the hard keys.

Move the Keyboard by touching and holding the translucent bar at the top and dragging it to the desired position. Close the keyboard by taping the X on the upper right corner or by pressing the KEYBOARD info key again.

6.8 Calculator

The CALC info key will open a new window with a calculator. Rather than being a Fly out window, the calculator is a separate window that can be moved around the screen. Open the scientific calculator by clicking on the CALC header bar. When a numeric input is required while programming, the Inc Set and Abs Set will set the calculated value into the field that is being programmed.

Move the Calculator by touching and holding the translucent bar at the top and dragging it to the desired position. Close the Calculator by tapping the X on the upper right corner or by pressing the CALC INFO key again.

7.0   DRO Mode

The ProtoTRAK RMX CNC operates in DRO Mode as a 3-axis digital readout. In addition to the large, easy-to-read dimensions, you have access to powerful capability that will make your manual work more productive than ever.

7.1   Status Bar in DRO Mode

The status bar, when in DRO mode displays the following field boxes. A sample of the status bar as displayed on the VMC2 is shown in Figure 7.1a and a sample of the status bar as displayed on the TMC's is shown in Figure 7.1b.

	ATC BACK	TOOL # 101		DIA 0.5000	Ctr Drill
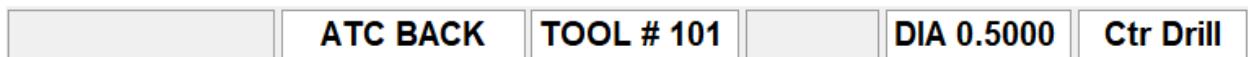
Figure 7.1a - VMC2 Status Bar in DRO Mode – Library tool in spindle.

	ATC LOC	1	TOOL # 1	LOC 1	DIA 0.5000	Ctr Drill
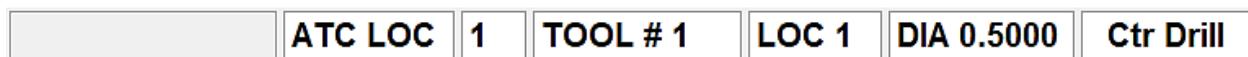
Figure 7.1b - TMC Status Bar in DRO Mode – ATC Tool shown.

VMC2 Status Bar Fields
Status Bar Box #1: Currently not being used.
Status Bar Box #2/3: ATC position
Status Bar Box #4/5: TOOL #/LOC # – Information about the tool in spindle. This information will change based on manual tool, library tool, and ATC tool along with ATC LOC. It will also display NO TOOL when there isn't a tool in the spindle. A highlighted yellow "TOOL#??" field may appear, which means the spindle sensor senses there is a tool in the spindle but it is not able to identify the tool information from tool table.
Status Bar Box #6: DIA – Lists the diameter of the tool.
Status Bar Box #7: Lists tool type.

TMC's Status Bar Fields
Status Bar Box #1: Currently not being used.
Status Bar Box #2: ATC LOC label
Status Bar Box #3: Location of the ATC relative to the tool change position (position straight across from spindle).
Status Bar Box #4/5: TOOL #/LOC # – Information about the tool in spindle. This information will change based on manual tool, library tool, and ATC tool along with ATC LOC. It will also display NO TOOL when there isn't a tool in the spindle. A highlighted yellow "TOOL#??" field may appear, which means the spindle sensor senses there is a tool in the spindle but it is not able to identify the tool information from tool table.
Status Bar Box #6: DIA – Lists the diameter of the tool.
Status Bar Box #7: Lists tool type.

7.2 DRO Functions

Once you press the DRO Mode key, you will encounter the following screen with tool status located at the top of the window:
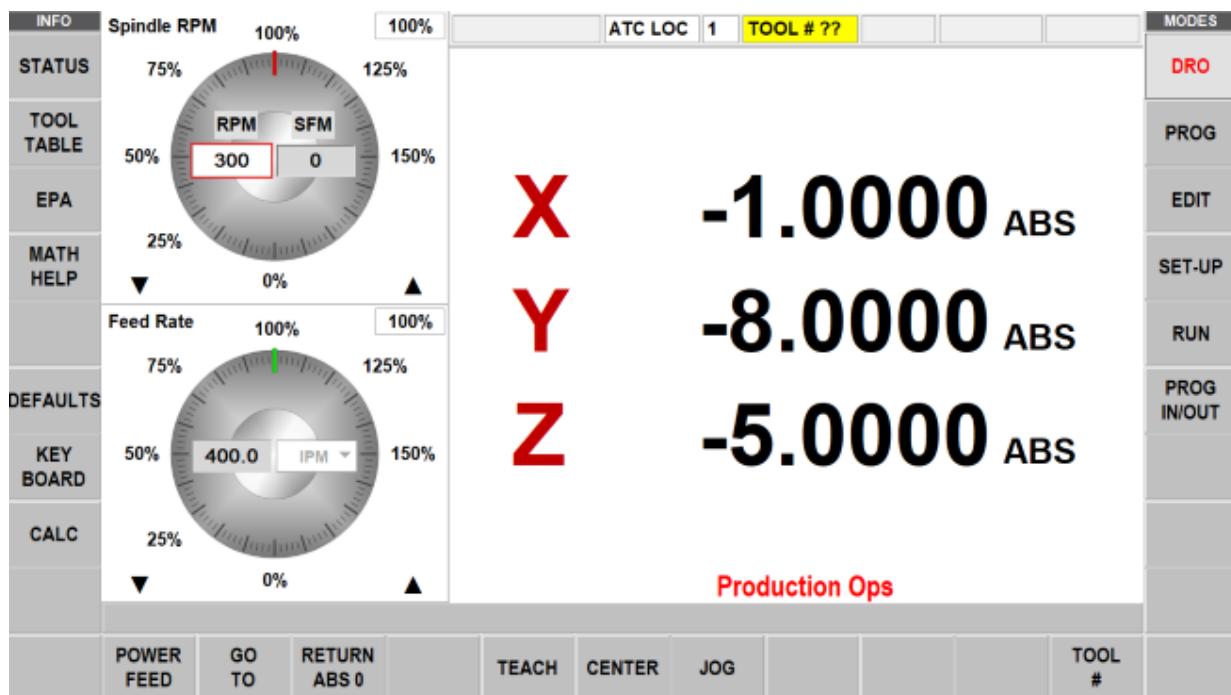
Figure 7.2 - The DRO Status bar shows carrousel location.

Production Ops/Tool Room Ops
When user switches between either operation, the operation in current use will listed at the bottom of the DRO screen in red font.

Inch to MM
To convert Inch to MM or MM to Inch tap the Status Info key and in Units at Start Up select Inch or MM. When you tap the Status key again the dimensions will be in the units you selected.

Selecting Dimension Reference
You may display your X, Y and Z dimensions in Incremental or Absolute for each axis. You may have a mixture of Absolute and incrementally dimensioned data.

To select between Absolute or Incremental Dimensions tap the ABS/INC label. The label and dimensions will toggle between Absolute and Incremental.

Resetting Dimensions
To reset, or zero out, Incremental Dimensions press the X, Y or Z hardkey and then the INC SET hardkey. The incremental dimension for that axis will be set to 0 (zero). Note, whenever you change your Absolute number, whether you reset or preset, you will reset (zero out) the Incremental Dimension.

To reset, or zero out, Absolute Dimensions press the X, Y or Z hardkey and then the ABS SET hardkey. The Absolute dimension for that axis will be set to 0 (zero). Please note that resetting the ABS Dimension will also reset the Incremental Dimension for that axis.

Presetting Dimensions

You can also enter preset dimensions in order to establish a reference from the current X, Y and Z position to some other point.

To preset an Incremental Dimension, press the X, Y or Z hardkey to select the axis. Enter the dimensional value and press INC SET. The incremental preset will have no effect on the Absolute dimension.

To preset an Absolute Dimension, press the X, Y or Z hardkey to select the axis. Enter the dimensional value and press ABS SET. Presetting an Absolute dimension will also reset (zero out) the Incremental dimension for that axis.

7.3   Apply Tool Library Data to DRO Operations

The ProtoTRAK RMX enables you to apply the tool offsets from your tool Library while you are working manually. This will save you from touching off tools that have already been set.

Note - *You may access the Tool Library at any time by pressing the Tool Table info key.*

1. Tap the TOOL # soft key.
2. Enter the desired Library number. The tool library number must be between 101 and 199.
3. The Z dimension will reflect the new Absolute dimension of the tool you entered. The Tool # at the top of the screen will display the tool number entered.
4. If you enter a tool # that doesn't exist in the Tool Library, no offset will be applied to the Z position.

   Note - *TOOL # soft key will not allow user to call ATC tools. When ATC tool is called, an error message will appear saying ATC operation is not allowed in DRO mode.*

7.4   Spindle RPM
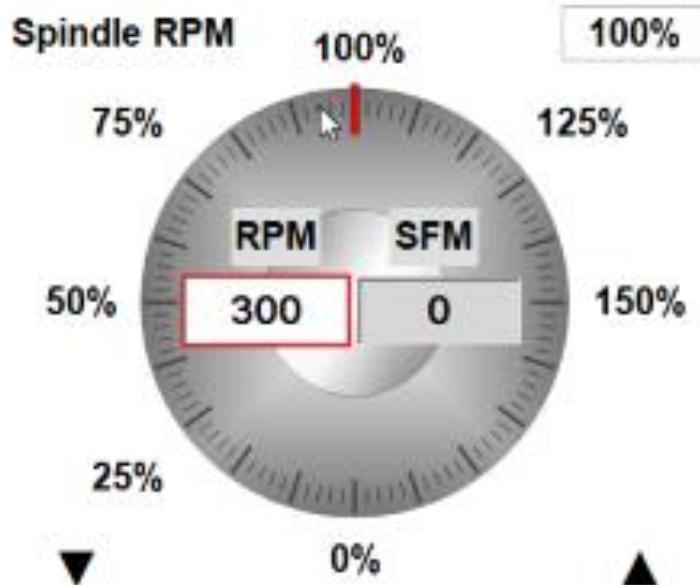
Figure 7.4 - The Spindle Speed control in the DRO Mode.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

7.4.1   Setting Spindle Speed Values

To Set Spindle Speed tap either RPM or SFM Spindle Speed box, enter the value up to the maximum and then press SET. The maximum RPM/SFM is set at the factory for the maximum rating of your machine model. The box highlighted in red indicates at which speed it will run and whether the program is run using RPM or SFM. In RUN Mode, the red outlined box toggles between RPM and SFM based on the spindle speed and values set in the program events.

7.4.2   Overriding Spindle Speed

You may override the set Spindle Speed from 0% of set up to 150%.

To override do one of the following:
- For fine adjustment, tap the up or down Fine Override arrow keys.
- Tap one of the set % values around the graphical dial, for example, 75%.
- Drag your finger in a circular motion along the diameter of the graphical dial as if you were physically turning it. If you move your finger out to a larger radius, you will notice that this still allows adjustment of the feedrate. It gives the user a finer control of the feedrate.

Note - *As you "turn" the dial with your finger, the dial will "stick" to 100%. To resume turning, briefly lift your finger and put it back down. We put in this slight sticking at 100% so that you can move your attention away from the screen while ramping the override up or down.*

As you apply overrides, the Spindle Speed and the Override % will update to the new values.

7.5   Feedrates in DRO Mode

You can set and override in order to do Power Feed, Go To or one of the other powered operations in DRO Mode.
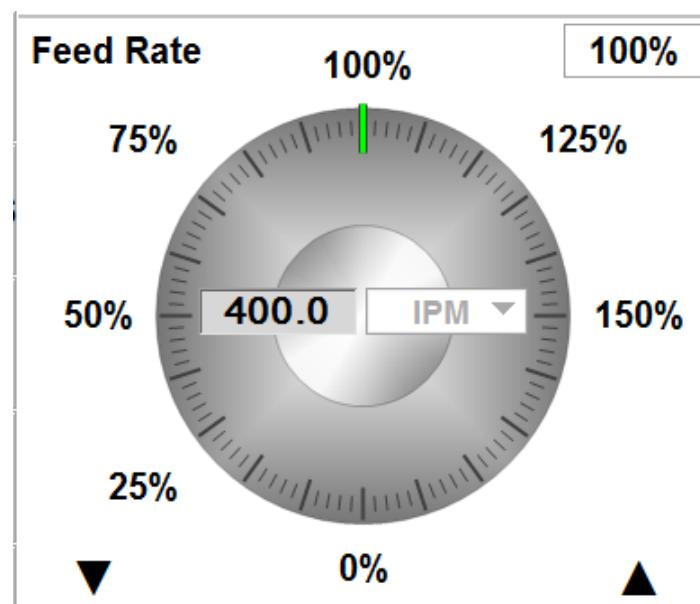
Figure 7.5 - The Feedrate control in the DRO Mode

7.5.1 Setting Feedrate and Units

To Set Feedrate tap the Feedrate box, enter the value and the press SET.

To change the Values between IPM (Inch or mm per minute) and IPT (Inch or mm per tooth) at Start up, tap Defaults and then Feed Moves.

7.5.2 Overriding Feedrates

You may override the set Spindle Speed from 0% of set up to 150%.

To override do one of the following:
- For fine adjustment, tap the up or down Fine Override arrow keys
- Tap one of the set % values around the graphical dial, for example, 75%.
- Drag your finger in a circular motion along the diameter of the graphical dial as if you were physically turning it.

Note - *If you lift your finger while turning, the dial will "stick" to 100%. To resume turning, lift your finger again first then it will respond when being dragged. We put in this slight sticking so that you can move your attention away from the screen while ramping the override up or down.*

As you apply overrides, the Spindle Speed and the Override % will update to the new values.

7.6 Power Feed

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

1. Press the POWER FEED soft key.
2. 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.
3. 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.
4. Do this for each axis you want to feed simultaneously. The message box will display what you've entered.
5. Initiate the power feed move by pressing GO.
6. The default Feed Rate is 10 IPM or 0.010 IPM (you can use IPT as well). Enter a new number in the feedrate box. While the machine is moving use the Feed Override to adjust it.
7. Press STOP to halt power feed. Press GO to resume.
8. Press RETURN soft key to return to manual DRO operation.

7.7 Go To

The Go To function in the DRO mode allows you to set an absolute 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 to 2.000" Absolute, you would input: Go To, X, 2, ABS SET. While the Go To window is displayed, the ProtoTRAK RMX will not let you pass that 2.000" dimension you set.

1. Press the GO TO soft key.
2. Enter the axis, X ,Y, Z, or all in turn.
3. Input the dimension(s).
4. Press Abs Set.
5. Crank the handwheel. Motion will stop at the entered dimension even if you continue to crank the handwheel.

7.8 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 the ram to Z tool change height 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 ram will not.

7.9 Center

The ProtoTRAK RMX will calculate the centers of lines and circles for you.
1. Press CENTER.
2. Select Line Center or Circle Center.
3. Follow the prompts in the message box on the screen.

The ProtoTRAK RMX will calculate the center from the information you've given it and display this calculation in the message box. If you desire, the control will feed at 100 ipm to the center location it calculated.

7.10 Jog

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

1. Press the JOG soft key. A flashing message will appear saying "CAUTION: JOG KEYS ARE ACTIVE".
2. To jog, press the X, Y or Z hard keys.
   Warning – *Motion will occur!*
3. To stop jogging, release the key.
4. The default feed is the maximum of 400 ipm.
5. Press the +/- hard key to reverse direction. When the number in the Feed rate box is negative, this indicates the minus direction.
6. Use the Feed Override functions to jog at a different speed.

7.11 Teach

Teach gives you the ability to create a program by manually mimicking the moves you wish for the CNC to make. 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.

Entering Teach Data

From the DRO Mode, press Teach. On the top of the screen, you will see an event counter.

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 soft keys 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 soft key 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 soft keys will cause the event counter to increase by one.

At any time, you may exit the Teach and return to the DRO screen.

Converting Teach Data to a Program

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.

So far, you have entered dimensions, but the ProtoTRAK RMX will need a little more information before you can make parts. The events you have defined with their X and Y dimensions are finished in the Program Mode.

8.0 Program Mode Part 1: Getting Started and Some General Info

8.1 Programming Overview

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

The basic strategy is to first fill in the initial program information in the Program Header screen and then program the features of the part by selecting the soft key event types (geometry) and then fill in the prompts one-by-one.

Enter the Program Mode by tapping the PROG Mode key. If you exit the Program Mode, the same screen displayed when you exit will be there when you return.

The ProtoTRAK RMX CNC 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.

To open a previously stored program, see Program In/out mode.

8.2 Info Keys

The ProtoTRAK RMX gives you extraordinary power and control over how your part is machined. As you are programming, you may use the Info Keys at any time:

- **Status:** Quick check of Program Name and other information.
- **Tool Table:** Check tool reference information or set up your tools as you program.
- **EPA:** Quick reference information on relevant topics to get you past some uncertainty about how to do something.
- **Math Help:** Convenient routines to help you calculate missing print data. Answers may be loaded directly into the program event.
- **Options:** Override Defaults or initiate additional capability that you may need on a particular event. If you find yourself frequently overriding a Default, you may want to consider changing the Default itself.
- **Defaults:** Change the programming style that you have defined on the fly to fit the current work. Any changes to Defaults will take place for the next event you select.
- **Keyboard:** Alphanumeric text and symbols for you to use to name the program or to enter in Engrave events (Advanced Features with Verify Option).
- **Calculator:** Basic calculator for quick math. Answers may be directly loaded into the event.

When you are finished with the Info Key, simply tap the key again to put the window away. Close the Keyboard and Calculator by tapping the X on the upper right corner of their window.

8.3 Program Header (Event 0) Screen

The first screen you see when you enter the Program Mode is the Program Header Screen.

The program name and general programming options you choose in the Program Header Screen will be summarized in the program as "Event 0".
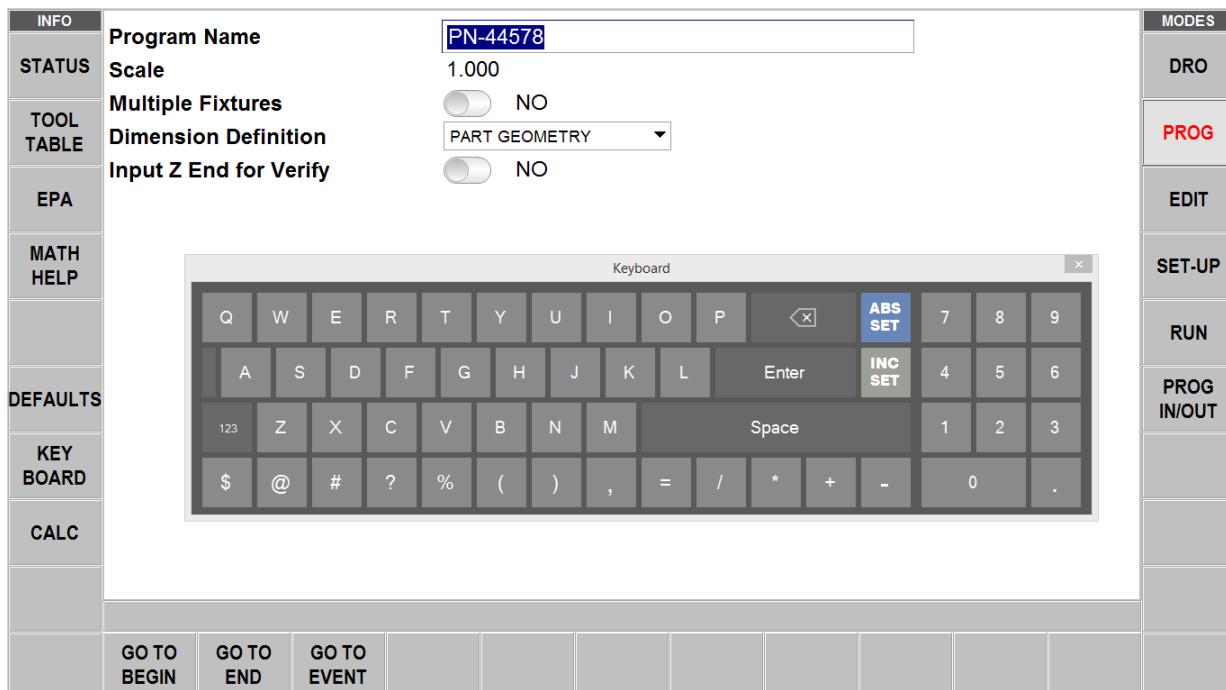
Figure 8.3 - The Program Header screen (event 0) with the Keyboard open for entering the part number.

Program Name: Data is entered using the keyboard or the Data Input hard keys. Press either SET after entering. Program names may be any length, but only 40 characters will be displayed in the field of the program name and up to 35 characters will be displayed in the Program Name field in the Status screen.

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.

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. See below for more explanation of how to use Multiple Fixtures. Multiple Fixtures is an Advanced Feature with Verify.

Dimension Definition: The ProtoTRAK RMX CNC gives you a choice in programming either tool path or geometry. Part Geometry programming allows you to define the geometry for your part and then the CNC does the 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 plane (see Section 5.3 for a definition of planes). For this reason, the ProtoTRAK RMX CNC 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. Otherwise, Part Geometry programming is assumed. Tool Path operates under the same rules as standard RS274.

A program must be entirely written in Part Geometry or Tool Path programming, you cannot combine the two methods in one program.

Input Z End for Verify: This prompt will appear only if you have the Verify option and you are writing a 2-axis part program. If you select Yes you will be prompted for Z End information in each event. This is needed to show the depth for each event on the solid model of the part you program.
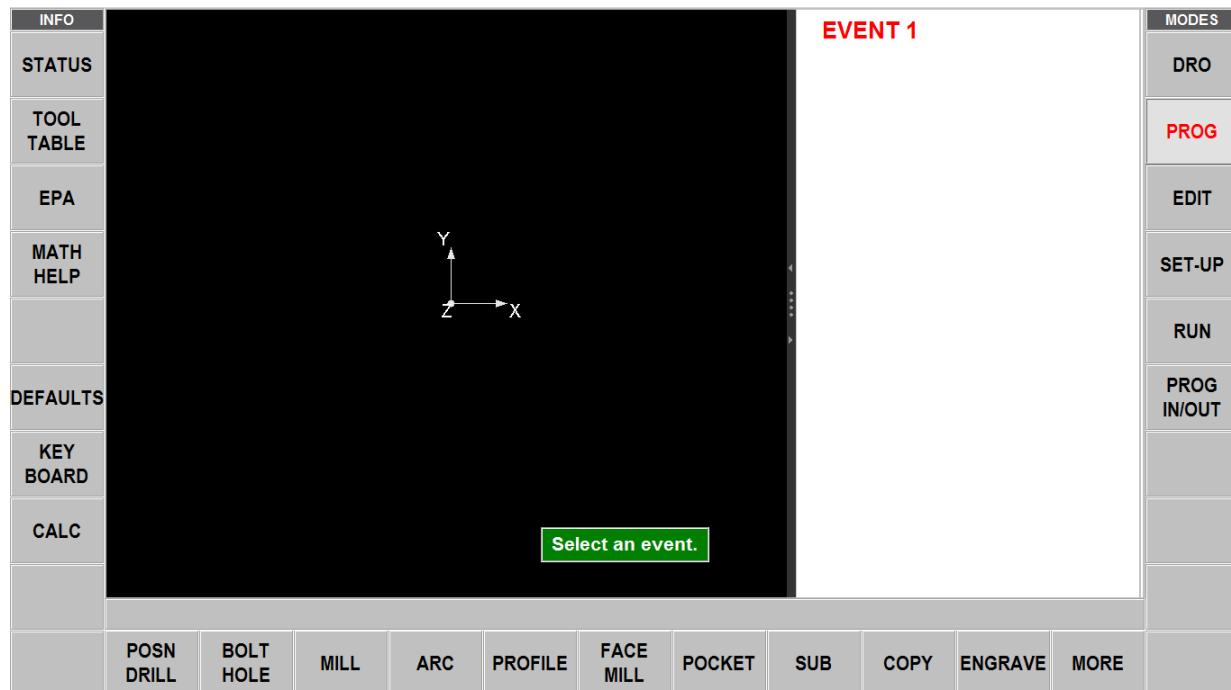
GO TO BEGIN: Takes you from the Header Screen to the first event of the program. You may also simply swipe to the left.

GO TO END: Takes you from the Header Screen to the last event of the program.

GO TO EVENT: Will prompt you to enter an event number, then takes you to that event when you press SET.

8.4 Starting to Program

To start programming, select Go To Begin or simply swipe the screen to the right to launch Event 1. At any time, you may move between the header and between events by swiping.
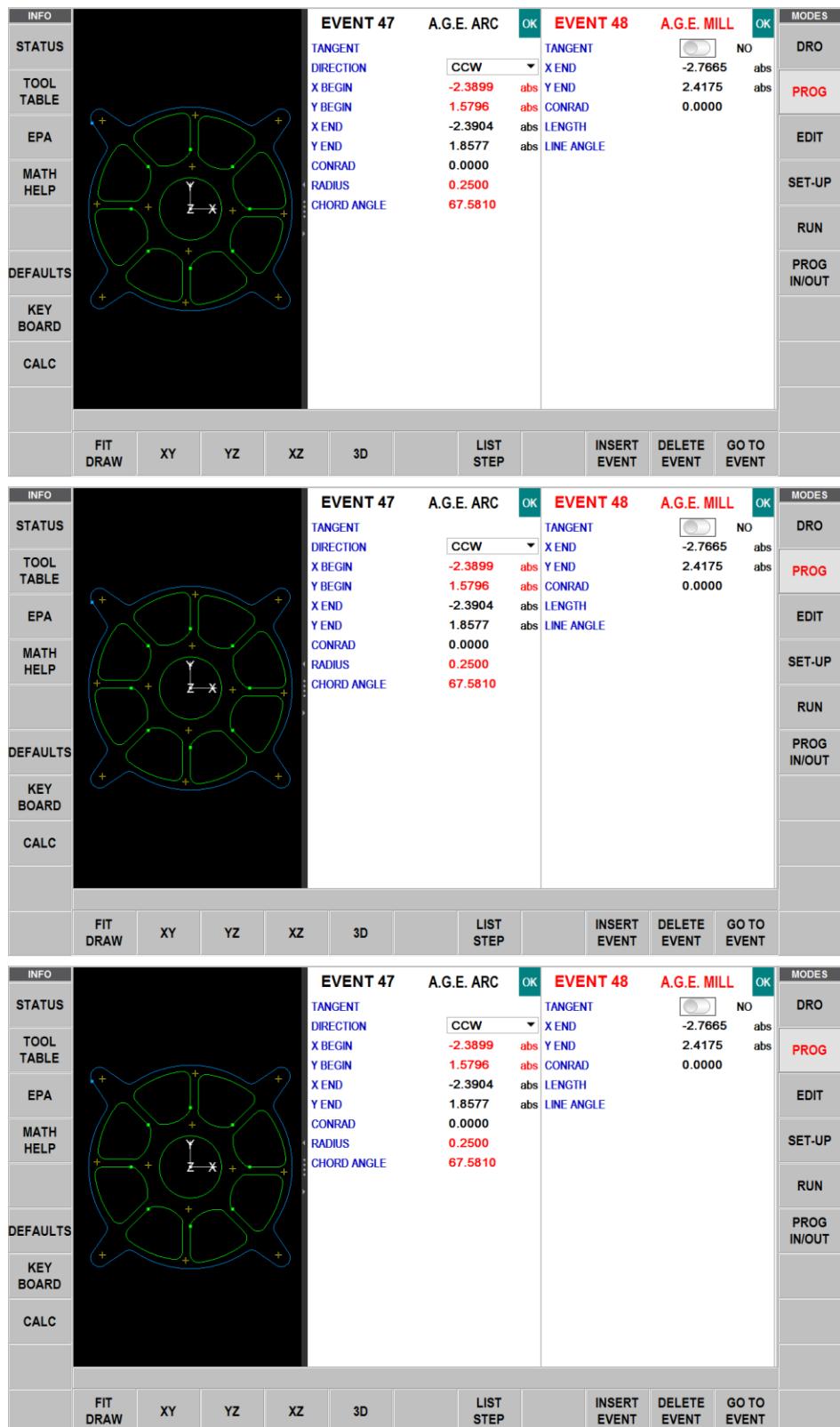

When an event is selected, all the prompts that need to be input will be shown on the right side of the screen. The first prompt will be highlighted. Input the dimension or data requested and press INC SET or ABS SET. For X, Y or Z dimension data it is very important to properly select INC SET or ABS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

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

8.5 Part Drawing While Programming

As you program, your inputs will be used to make a drawing of the part to show you what you've programmed so far. You may display one or two events by swiping the right edge of the border of the drawing and the drawing will automatically resize to the amount of screen left to it.

Figures 8.5 - Swipe the screen to select between views of the events and drawing

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

8.6   Soft Keys Within the Event

Program Events will be described in the next section.

Once an event geometry is selected, the following Soft Keys will appear.

FIT DRAW	XY	YZ	XZ	3D		LIST STEP		INSERT EVENT	DELETE EVENT	GO TO EVENT

Figure 8.6 - Soft Keys within an event.

FIT DRAW: Resize the drawing to fit the screen after you have zoomed or pinched it.

XY: XY Plane view of part

YZ: YZ plane view of part

XZ: XZ plane view of part

3D: 3D view of part

LIST STEP: Launches a process to show the part geometry as related to the Event type and number.

When the LIST STEP key is selected, a list of the Event numbers and types appear on the right side of the screen and additional Soft Keys appear at the bottom:
- **STEP FWD:** Each key press moves you up in event number. As this occurs, the drawing will highlight the geometry in the event by turning to purple and the event number will appear in the upper left of the drawing screen.
- **STEP BACK:** Moves backward through the drawing.
- **START AT EVENT:** Jumps you to the Event number in the drawing that you input.
- **RETURN:** Takes you back to the previous soft key selections and screen view.

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: Deletes the event on the right side of the screen.

GO TO EVENT: Jumps you to the event you input.

8.7   Z Rapid and Safety Plane

Many events will give you the option of turning on and programming a Z Safety Plane. The Z Safety Plane is a dimension in Z Absolute that you define. It is there to allow you to program a Z move needed to clear an obstruction.

For example, say you had Mill event as Event 1 and a Pocket event as Event 2, and on the part there is a clamp between the end of Event 1 and the beginning of Event 2. The normal toolpath at the end of Event 1 would be:
- Z moves to the higher of Z rapid for Event 1 and Event 2
- XY moves at rapid to the beginning of Event 2.
- Z moves to Z rapid of Event 2, then to XYZ beginning of Event 2

This direct path between events would cause the tool to hit the clamp. In order to avoid this, you could enter a Z Safety plane within event 1.

The toolpath would then be:
- Z moves to the Z Safety Plane set in event 1 before going to Event 2
- XY moves at rapid to the beginning of Event 2.
- Z moves to Z rapid of Event 2, then to XYZ beginning of Event 2

8.8   Editing Data While Programming

In addition to the List Step view, The ProtoTRAK RMX CNC allows you to move easily though your program by swiping the events back and forth. Previously programmed prompts and events may be easily edited. With your event on the right side of the screen, tap the data that you wish to change or replace. Simply enter the desired data and press INC SET or ABS SET. The ProtoTRAK RMX will adjust the program and update the drawing.

8.9   Finish Cuts

When applicable for the event geometry, a Finish Cut prompt will appear. If a value is entered, the prompts for Finish Feedrate, Finish RPM and Finish Tool # will remain. If a 0 is entered, indicating no finish cut, those prompts will disappear.

When a finish cut is programmed, the ProtoTRAK RMX will first rough out the material, leaving the amount of material specified in the Finish Cut prompt. This material will be machined at the programmed Finish Feedrate and RPM and with the programmed Finish tool.

A default value for finish cuts for Pockets, Profiles and Islands may be set in the Defaults Info Key.

8.9.1   Z Finish Cut

Use finish (rest tool) on Z finish cut for Pockets and Islands.

8.10   Programming in 2 vs 3 Axis

The ProtoTRAK RMX may be programmed in either 2 or 3-axis in Production Ops and only 2-axis in Tool Rooms Ops. Each program must be entirely in either 2 or 3-axis. When programming in 2 axis, the prompts for Z will not appear.

You can change your Mode between 2 and 3 axis with the Status Info key. When you change, the screen will revert to the Start Up screen.

8.11   Using Multiple Fixtures

When you want to use multiple fixtures, set the Multiple Fixtures setting in the Program Header screen to Yes.

You may run your program using up to six fixtures with the first fixture being the base fixture. 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 uses fixture number one. It doesn't have to; we just believe it is clearer that way.

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

8.11.2   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 12.2 for setting up fixture offsets.

9.0 Program Mode Part 2: Program Events

Events are fully defined pieces of geometry. By programming events, you tell the ProtoTRAK RMX CNC 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.

In the ProtoTRAK RMX, defining your event is made even easier by the interaction of the Defaults you may set to define your programming style and the Options that enable you to change defaults or have additional functions.

Please see section 6.5 for information about Defaults and 6.6 for information about Options.

9.1 Prompts Found in Events

The following is a description of the Prompts you'll encounter after selecting an Event. Only the prompts appropriate to the event you select will appear.

 # Holes: The number of holes in the bolt hole pattern

 # Passes: The number of cycles to machine to the final depth spaced equally from Z Rapid to Z End (**Hint -** *Keep Z Rapid small*).

 # PECKS: The default is displayed either Variable, Fixed or Chip Break. You may override the default by choosing the Options Info key.

 # Repeats: The number of times the programmed events are repeated in the Subroutine events.

 # Rest Passes: The number of cycles to machine to the final depth in the rest (residual) material with the finish tool.

 # Rev: The number of revolutions in the helix.

% Feed: The percentage of the feeds in the programmed events to apply to the Subroutine events.

% RPM: The percentage of RPM in the programmed events to apply to the Subroutine events.

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

Comments: Select to turn On/Off for all events. If you select YES, 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. Mill and A.G.E. Arc event. Comments may be composed of letters, numbers and some symbols and may be up to 20 characters.

Conrad: The dimension of a tangential radius to the next event (that must lie in the same plane for part geometry programming). May also be the corner radii of a rectangular pocket.

Continue: On/Off selection to ask user if they want to continue running the program after the event.

Cutting Order: Forward – The toolpath is from the first event to the last event. Backward – The toolpath is from the last event to the first event. 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.

Direction: The clockwise or counterclockwise 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.

FEED PER MIN: The Feedrate in Inch per minute or mm per minute.

FEED PER TOOTH: The Feedrate in inch or mm per tooth. The number of flutes must be defined for the tool in the Tool Table.

FIN CUT: The width of the finish cut. If 0 is input there will be no finish cut. You will not receive this or the other Finish prompts if the TOOL OFFSET is Tool Center.

FIN FEED PER MIN: The finish cut milling feedrate in in/min from .1 to 400 (250 for Z), or mm/min from 5 to 10160 (6350 for Z).

FIN FEED PER TOOTH: The Finish cut feedrate in inch or mm per tooth. The number of flutes must be defined for the tool in the Tool Table.

FIN RPM: The spindle RPM for the finish cut.

FIN SURFACE SPEED: Finish Surface Speed in Surface Feet per Minute, which is the speed of the outside Diameter of the tool.

FIN TOOL #: Tool number you assign for the Finish Cut.

First Event #: The event number of the first programmed event for the subroutine.

Height: The height of each letter or number being engraved.

Last Event #: The event number of the last programmed event of the subroutine.

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).

Mirror Axis: The selection of the axis or axes to be mirrored (input X or Y or XY, SET) in the Subroutine Mirror Events.

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 / 20 = .05 inches

Radius: The finish radius of the arc circle or the bolt hole pattern.

RPM / SFM: The default is displayed either RPM or SFM. You may override the default be choosing the Options Info key.

Side: Inside for an ID thread, outside for an OD thread.

Spindle Speed: Programmed as either **RPM** or **SFM** (Surface Feed).

Text: Enter the text you want to engrave and press SET.

Tool #: The tool number you assign. SET without a number will use the tool number of the previous event.

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

X Begin: The X dimension to the beginning of the mill or arc cut.

X Center: The X dimension to the center of the arc or bolt hole pattern; incremental is from X End.

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

X Offset: The offset value for x-axis in a subroutine event.

X: The X dimension to the hole.

X1: The X dimension to the first diagonal point of a rectangle.

X3: The X dimension to the opposite diagonal point of a rectangle.

XYZ FEED PER MIN: The milling feedrate in inches or mm per minute.

XYZ FEED PER TOOTH: The XYZ milling feed rate per tooth in inches or mm. This is the chip thickness.

Y Begin: The Y dimension to the beginning of the cut.

Y Center: The Y dimension to the center of the arc or bolt holt pattern; incremental is from Y End.

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

Y Offset: Offset value for y-axis in a subroutine event.

Y: The Y dimension to the hole.

Y1: The Y dimension to the first diagonal point of a rectangle.

Y3: The Y dimension to the opposite diagonal point of a rectangle.

Z Begin: The Z dimension to the beginning of the cut.

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

Z End: The Z dimension to the end of the cut; incremental is Z Begin.
Z FEED PER MIN: The drilling or vertical milling feed rate in inch or mm per minute

Z FEED PER TOOTH: The Z direction milling feed rate per tooth in inches or mm. This is the chip thickness.

Z Rapid: The Z dimension at which the Z stops rapid and begins to feed at the programmed Z feed rate.

Z Safety Plane: A dimension in Z Absolute that you define. It is there to allow you to program a Z move needed to clear an obstruction and ensure that every tool will not accidently contact fixtures that stick up above the part.

9.2   Event Types
9.2.1   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.

9.2.2   DRILL TAP Event

This will drill, bore or tap a hole in the XY location that you enter.

Press the DRILL TAP soft key.

Drill, Bore, Tap, Helix: selects whether the hole is to be drilled, bored, tapped or machined in a Helix.
- **Drill** - Selecting this soft key positions the table to the specified X and Y position, moves the HEAD at rapid to the Z RAPID location, feeds to the Z END location, and rapids back to Z RAPID.

- **Bore** - Selecting this soft key positions the table to the specified X and Y position, moves the HEAD at rapid to the Z RAPID location, feeds to the Z END location, and feeds back.

- **Tap** - Selecting this soft key rapids to the top of the part and starts to tap down, then it reverses the spindle and taps out. Tap events allow you to tap holes with a rigid holder. The feedrate of the thread will be calculated from the pitch and RPM entered.
    - Prompt in the Tap Event – **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

Tapping Notes and Recommendations
- Harder materials will require slower speeds and tap size may also be limited.
- Make sure your tap is not dull. Dull taps will require more torque to cut and may not cut threads to specification.
- Cutting oil will play a large role in determining the size of tap you can use in a given material.
- Make sure your tap is running true in the holder.

Helical

This positions the cutter at the hole Diameter (offset by cutter diameter) at Z Rapid. From there the cutter moves in a helical motion in X, Y and Z feeding at the Pitch entered until it reaches Z End. At Z End, the cutter will move in X and Y to clean up the bottom of the hole. To finish, the cutter will feed to the center of the hole and then move at rapid feedrate out of the hole.

Additional Prompts in Helix:
- **Pitch** - Input in inches (mm) 1 divided by the number of X, Y, Z revolutions per inch (mm). For example, if you wanted 20 revolutions per inch you would enter .05 (1 ÷20).
- **Diameter** - Diameter of the hole.

9.2.3   Bolt Hole

This will drill, bore or tap a series of holes evenly spaced around a center you define.

Bolt Hole Drill

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

Bolt Hole Bore

This positions the table to the specified X and Y position, moves the HEAD at rapid to the Z RAPID location, feeds to the Z END location, and feeds back.

Bolt Hole Tap

This positions the table to the specified X and Y position, moves the HEAD at rapid to the Z RAPID location, coordinates the spindle and feed at Z Begin, feeds to the Z END location, and feeds back. See section 9.2.17 for more details.

Helical

This positions the cutter at the hole Diameter (offset by cutter diameter) at Z Rapid. From there the cutter moves in a helical motion in X, Y and Z feeding at the Pitch entered until it reaches Z End. At Z End, the cutter will move in X and Y to clean up the bottom of the hole. To finish, the cutter will feed to the center of the hole and then move at rapid feedrate out of the hole.

Additional Prompts in Helix:
- **Pitch** - Input in inches (mm) 1 divided by the number of X, Y, Z revolutions per inch (mm). For example, if you wanted 20 revolutions per inch you would enter .05 (1 ÷20).
- **Diameter** - Diameter of the hole.

See the section 9.2.17 for more information about tapping on TRAK Machining Centers.

9.2.4   Mill

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).

9.2.5 Arc

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. 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 180°, 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 RMX CNC 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.*

9.2.6 PROFILE Events

These events allow 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 RMX CNC will automatically initiate the powerful Auto Geometry Engine. See Section 10.0 for more information about programming with A.G.E.

Finish cuts and the number of passes are automatically programmed for Profile events if these values are set in the Default Info key. Finish Cuts are only programmed when the Tool Right or Tool Left positions are selected.

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

Rectangular Profile
Press the RECTANGLE soft key if you wish to mill a rectangular frame with or without corner radii (all corners are 90° right angles).

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 initiates the powerful Auto Geometry Engine to define a shape made up of straight lines (Mills) and arcs. See section 10 for more information about programming with the powerful AGE feature.

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 machining information that applies to the entire profile.

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 10.0.

9.2.7 Face Mill

The Face Mill event will cut a rectangular area you define by opposite corners. The toolpath for the Face Mill Event should start from the X BEG/Y BEG corner and cut in the X +/- direction. It always starts off the area you define and machines the top of the part using the Step Over Percentage and the Cutting Method you define in the Defaults. User can select the cutting order.

9.2.8 Pocket Events

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.

Circle Pocket

The Circle Pocket will machine the circumference defined and the material on the inside. You may place Islands inside a circle pocket. Two tool path pattern options available under circle pocket are:
- Offset
- Spiral (Under the Z entry method, Helical will not work for spiral tool path.)

Rectangular Pocket

The rectangular Pocket will mill a rectangular shape with all corners 90-degree right angles and the sides parallel to the X and Y axes and the material on the inside. You may place Islands inside a rectangular pocket.

Irregular Pocket

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. A Conrad may be added to the last event of an Irregular Pocket and it will be inserted between the end of the last event and the beginning of the first event.

9.2.9   Island Events

Island Events are programmed by first selecting Pocket and then 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 pocket. You may place Islands inside an irregular pocket event.
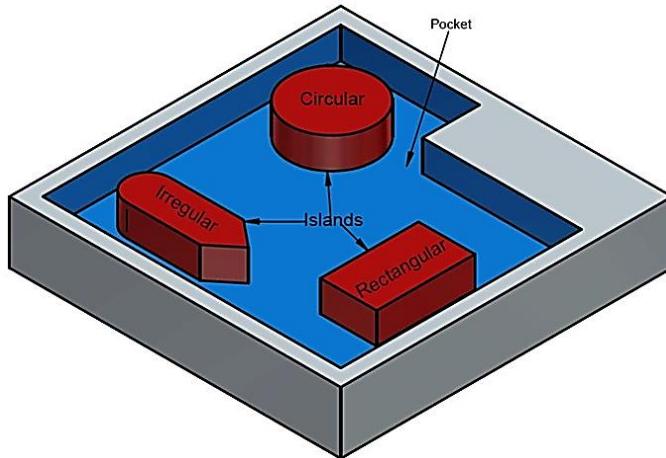
Figure 9.2.9 *Islands of any shape may be programmed within a pocket.*

This powerful feature gives you an easy way to do complicated shapes. In order to organize it properly, the ProtoTRAK RMX guides you through a sequence that helps keep things clear as you go.

First, from the Select an Event screen, you select POCKET.
Next, select ISLAND.
Next, you will program the Pocket in which the island will sit.

Once the Pocket is defined, you will have a choice of Island shapes:

CIRCLE ISLAND: The circumference defined.

RECTANGULAR ISLAND: A rectangular shape with all corners 90-degree right angles and the sides parallel to the X and Y axes.

IRREGULAR ISLAND: Initiates the powerful Auto Geometry Engine for you to define a shape made up of straight line and arcs.

END ISLAND: Tells the ProtoTRAK that you have no more islands to place inside the pocket. You can use finish (rest tool) on Z finish cut for pockets and islands.

9.2.10   Subroutine Events

The Subroutine Events are used for manipulating previously programmed geometry within the XY plane.
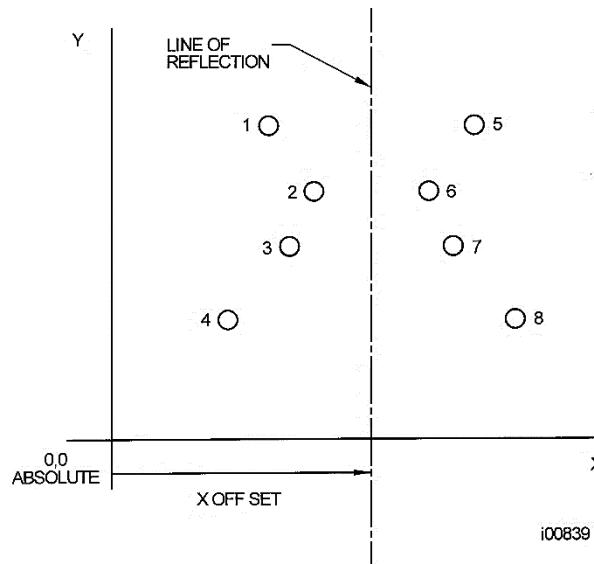
The Subroutine Event is divided into four options: Repeat, Mirror, Rotate, and Sub Part.

Repeat and Rotate may be connective. As long as the rules of connectivity are satisfied you may add 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 below:

Figure 9.2.10a - Holes 1-4 are mirrored across the Y axis to 5-8, respectively, about a line X OFFSET from X=absolute 0.

CUTTING ORDER:
- Forward – the toolpath is from the first event to the last event.
- Backward – the toolpath is from the last event to the first event.

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; however, you can rotate a mirror event. Consider the figure below:
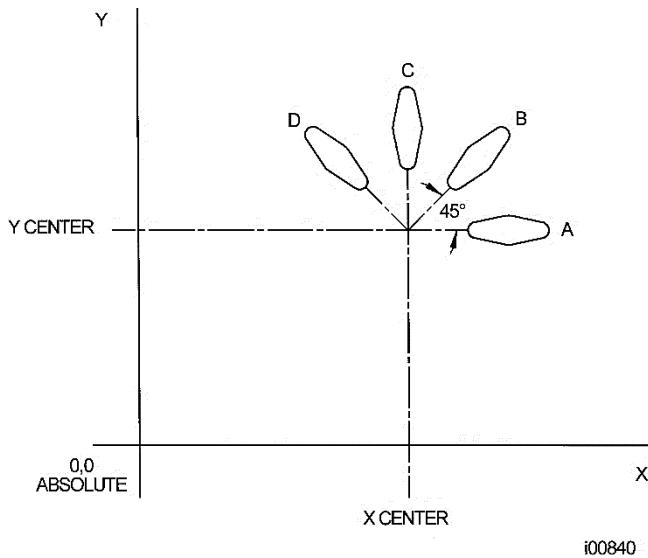
Figure 9.2.10b - Shape A programmed with 4 MILL events and Conrads. Using ROTATE, these 4 events are rotated through a 45-degree angle about a point offset from absolute zero by X Center and Y Center dimensions. A is rotated 3 times to produce shape B, C, and D.

Sub Part

REPEAT PART event is used to make multiple numbers of the same parts. This event has the following fields – First Event #, Last Event # and # of Parts. You cannot have more than one sub part event in program and this event will not apply for GCD programs or when you use multiple fixtures in your program. You can run this event by using the tool-by-tool option.

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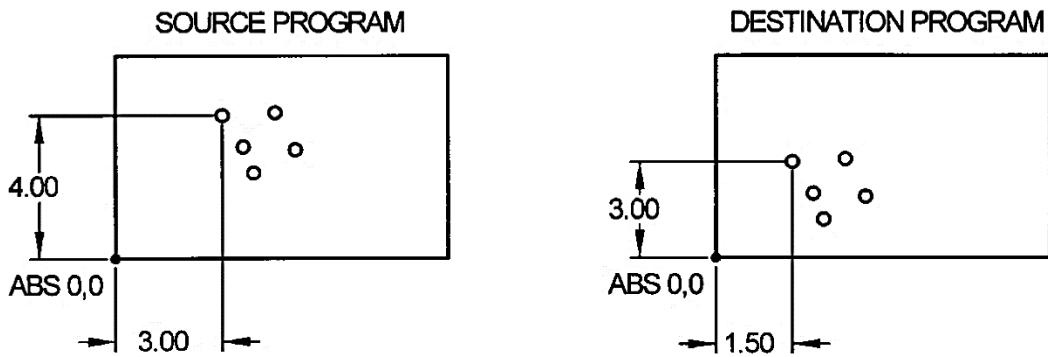
Copy Events are programmed exactly like Subroutine Events. The only difference is that in Copy the events are rewritten into subsequent events. If, for example, in event 11 you Copy Repeated events 6, 7, 8, 9, 10 with 2 repeats, events 6-10 would be copied with the input offsets into events 11-15, and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

Copy Events may be Repeat, Mirror, Rotate or Drill to Tap.

Copy is very useful. With Copy you can:
- Edit the events that are being repeated, mirrored or rotated without changing the original events.
- Connect 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, then delete the original.
- Convert a series of drill events to tap events.

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 11.6.

SOURCE PROGRAM   **DESTINATION PROGRAM**

4.00   3.00

ABS 0,0   ABS 0,0

3.00   1.50

i00965

Figure 9.2.11 - In the above example, the offset that puts the
group of holes in the desired location is X=-1.50 and Y=-1.00.

9.2.12   Helix Event

The Helix event will allow you to precisely define a helical tool path.

Note - *There is a separate Thread Mill Event for thread milling. You may quickly and easily program your cutter to enter Pockets and clear off events in a helical move. Simply set the Default Z Entry Method to Helical.*

9.2.13   Engrave

The Engrave Event allows you to machine numbers, letters and special characters as part of a part program.

When programming with the Engrave Event, the ProtoTRAK will construct a box to contain the text you define. This box may be referenced by the location of its lower left corner, or by its center. In addition, the box may be presented in a radius or mirrored.

Prompts in the Engrave Event:
- **X and Y Begin:** Define the lower left corner (for the Horizontal orientation) or the center of the box of text (for the Vertical or Radial Orientation).
- **Z End:** The Z dimension at which the Z stops rapid and begins to feed at the programmed Z feedrate.
- **Height:** The height of each letter or number.
- **Text:** Tap the Keyboard Info Key and enter the text you want to engrave and press SET.
- **Z FEED PER MIN:** The milling feed rate in inch or mm per minute
- **XYZ FEED PER MIN:** The milling feedrate in inches or mm per minute
- **XYZ FEED PER TOOTH:** The XYZ milling feed rate per tooth in inches

Additional functions in Engrave are enabled by tapping the Options Info Key.
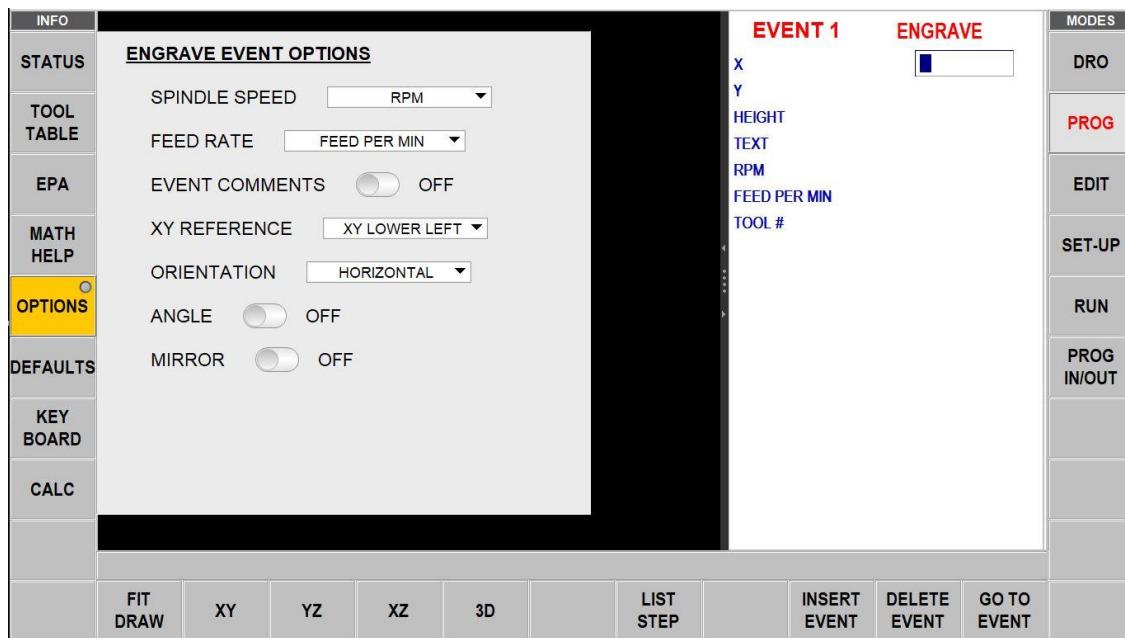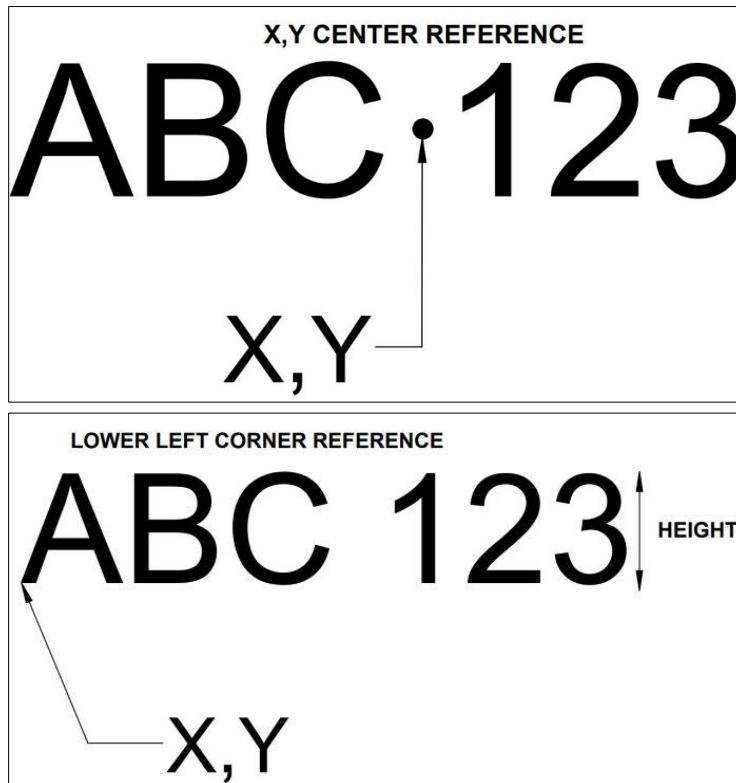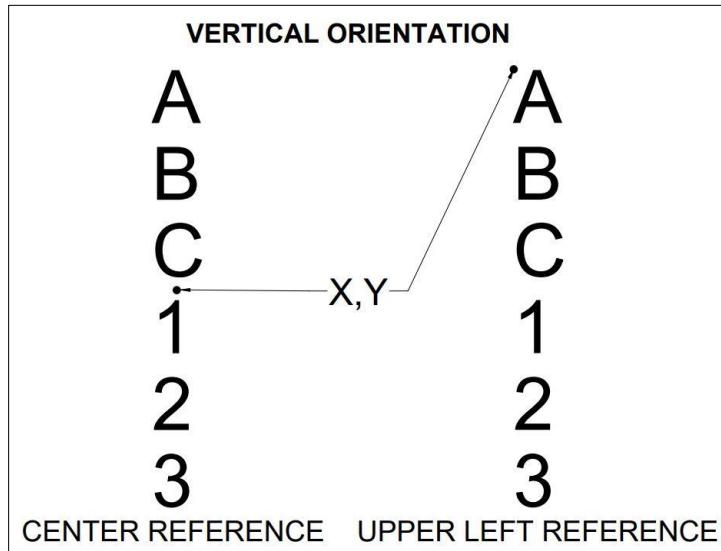
Figure 9.2.13 - RMX Screen Listing Engrave Options

Options Unique to the Engrave Event:
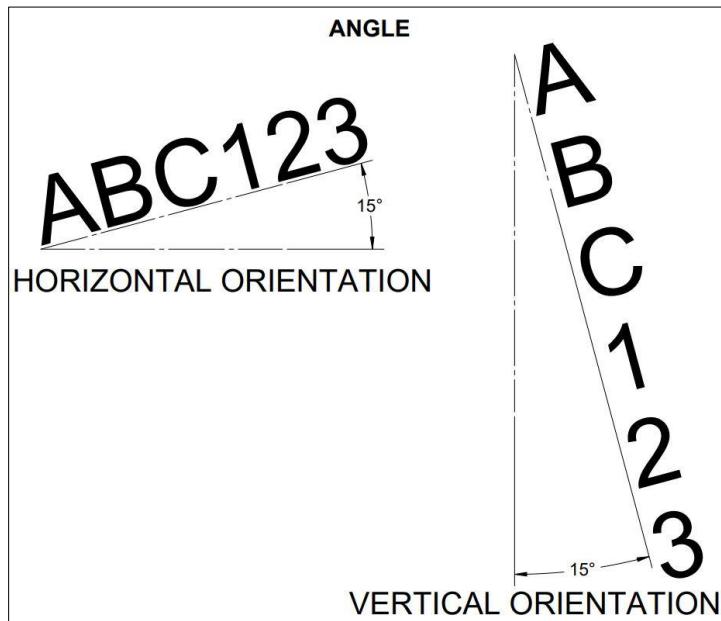
- **XY Reference:** Will use the values you enter to define the lower left corner of the text box or its center as shown below.

- **Orientation:** The text box may be Horizontal (like the text in this manual), Vertical (up and down) or Radial (around the axis you input).
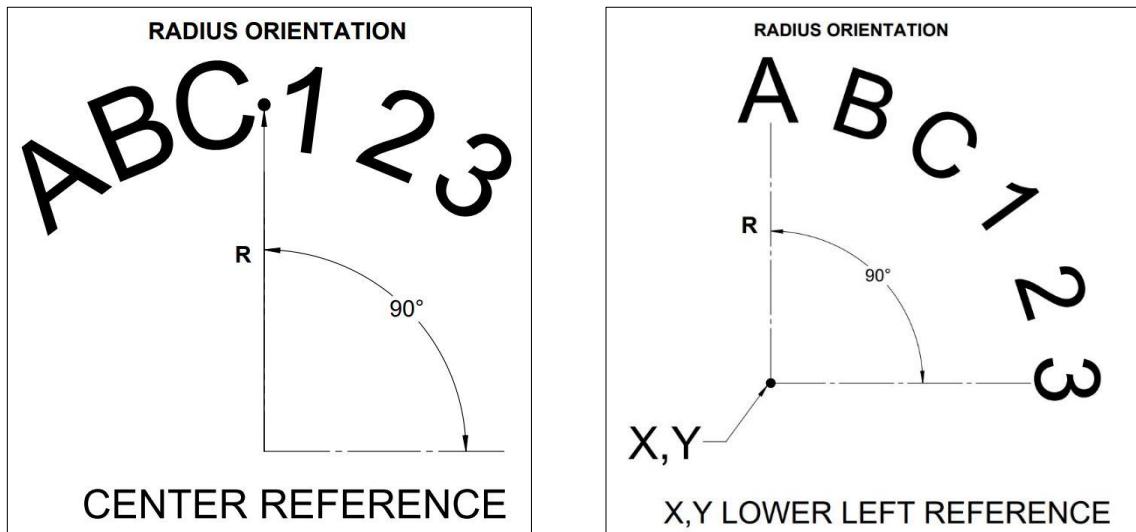
- **Angle:** Will work with either the Horizontal or Vertical Orientation.

- **Mirror:** Will Mirror the text about the X, Y or XY Axis defined by the X and Y inputs. The mirrored text will replace the original text and occupy the same place on the part.

Additional Prompts When Radius Orientation is Selected:

- **Radius:** The radius value you enter will apply with the center being the X and Y inputs and the bottom of the text box.
- **Radius Angle:** The angle from 3 o'clock to the beginning of the text box.

RADIUS ORIENTATION

CENTER REFERENCE

RADIUS ORIENTATION

X,Y LOWER LEFT REFERENCE

9.2.14 Thread Mill Event

Thread Mill event enables you to easily program an ID or OD thread.
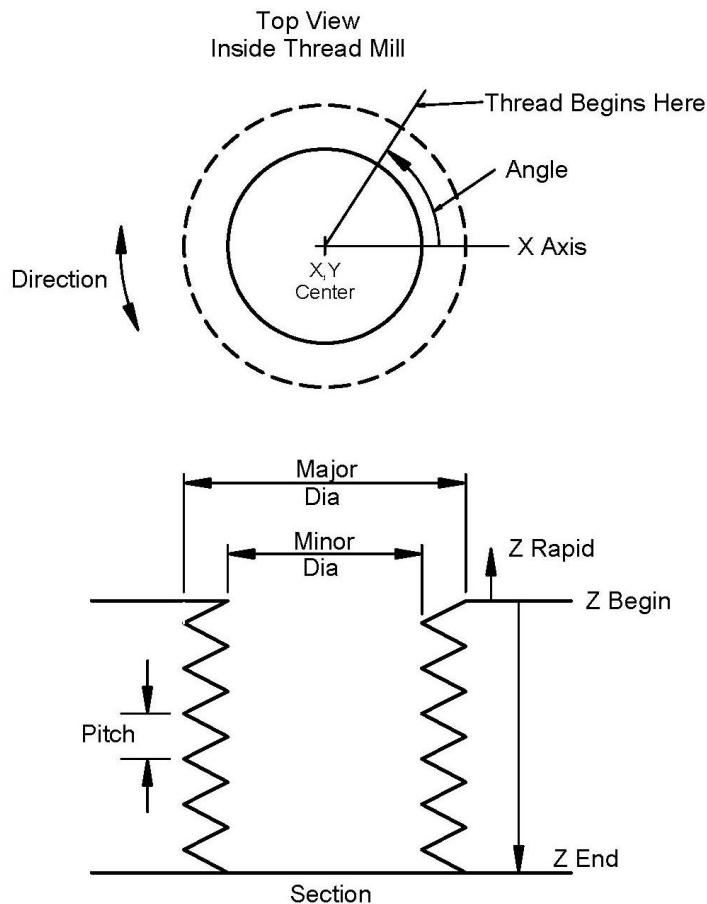
Figure 9.2.14 - Thread Prompts

Special prompts in the Thread mill event:

- **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 / 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:** Inside for an ID thread, outside for an OD thread.
- **ANGLE:** The angle the tool feeds into the beginning depth, where the thread begins
- **DIRECTION:** Clockwise or counterclockwise
- **# PASSES:** The number of passes to cut the thread to its final depth

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

9.2.15 Aux Events

Aux Functions allow you to control:
- Coolant
- Air
- Pulse to interface to a programmable indexer (O)
- Auxiliary Output that is used to interface to some secondary device (O)
- Part Change Position
- Coolant through Spindle (CTS)

For example, if you wish to have the coolant start when a milling event starts and then turn off at the end, simply insert an Aux event with Coolant On before the milling event and then another with Coolant Off afterwards.

The COOLANT, CTS, and AIR buttons on the Run Panel must be in AUTO for coolant, CTS, and air to work in your program.

Note - *The coolant, CTS, and air will turn off automatically at all tool changes and then back on automatically when the program is resumed.*

The CTS will only function while in Tool Room Ops/Production Ops, DRO mode/RUN mode, or TRAKing/CNC RUN when the door is locked, because its high pressure may cause coolant to spray everywhere.

When Part Change Position is turned on, the tool table moves forward in y-axis for easy accessibility to switch parts.

9.2.16 Pause

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 stop the program run, turn off the spindle (if you choose, you could leave it on) and move the head to the Z rapid location with the X and Y position corresponding to the end of the previous event.

Pause events are useful if you want to stop the program to make a measurement, change a fixture, etc.

Note - *In general, you should avoid programming a PAUSE event between two connective events. 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. You can also choose to leave the spindle on. The default is to turn it off.

In run, restart the spindle and press the GO key after a pause to continue.

9.2.17   Clear Off Events

Clear Off Event is an option that user can purchase.

Clear Off Events are programmed by first selecting Profiles and then stock.

A Clear Off event allows users to clear off material from around closed profiles (islands) or open profiles. First the user needs to define the profiles and then define some stock (boundary) as to what material to remove. Roughing will remove material from outside toward inside and tool will start off the part.
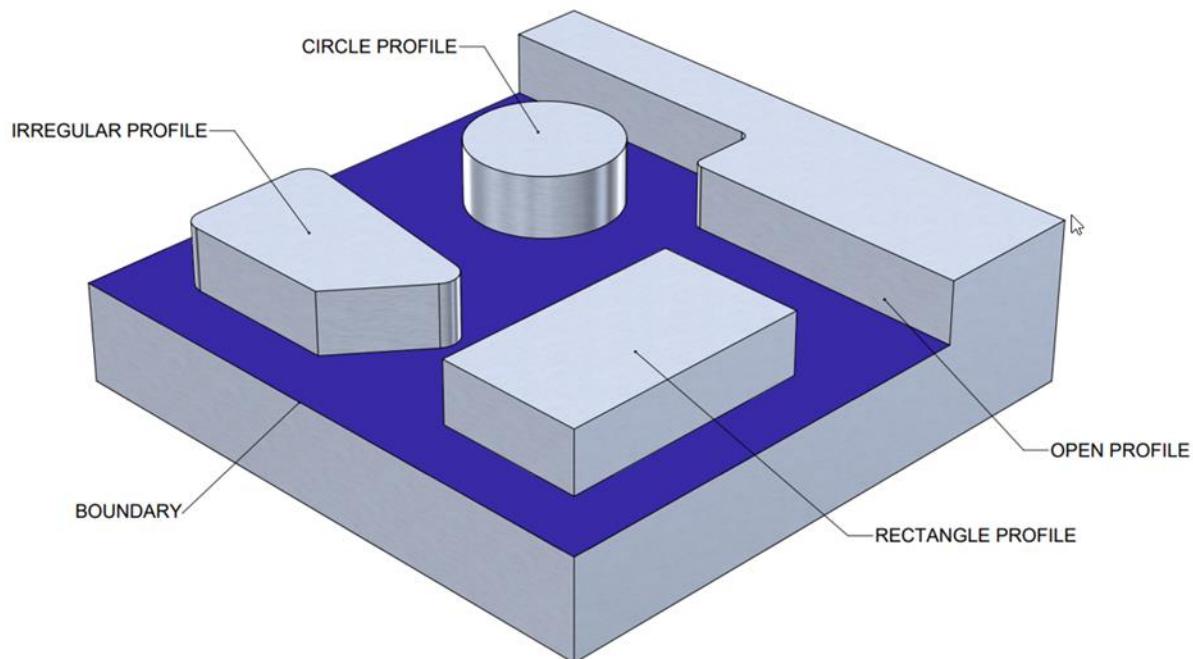
Figure 9.2.17a Materials can Clear off around both open and closed profiles

This powerful feature gives you an easy way to do complicated shapes. In order to organize it properly, the ProtoTRAK RMX guides you through a sequence that helps keep things clear as you go.

First, from the Select an Event screen, you select CLEAR OFF.
Next, select Circle Profile or Rectangle Profile or irregular Profile.
Next, you will program the Profile around which Clear Off will happen.
You can create 1 or more profiles using the above 2 steps. Note: the tool offsets for all the profiles inside a Clear Off sequence should be same. You will get the below message when you start creating 1st profile in Clear Off sequence.
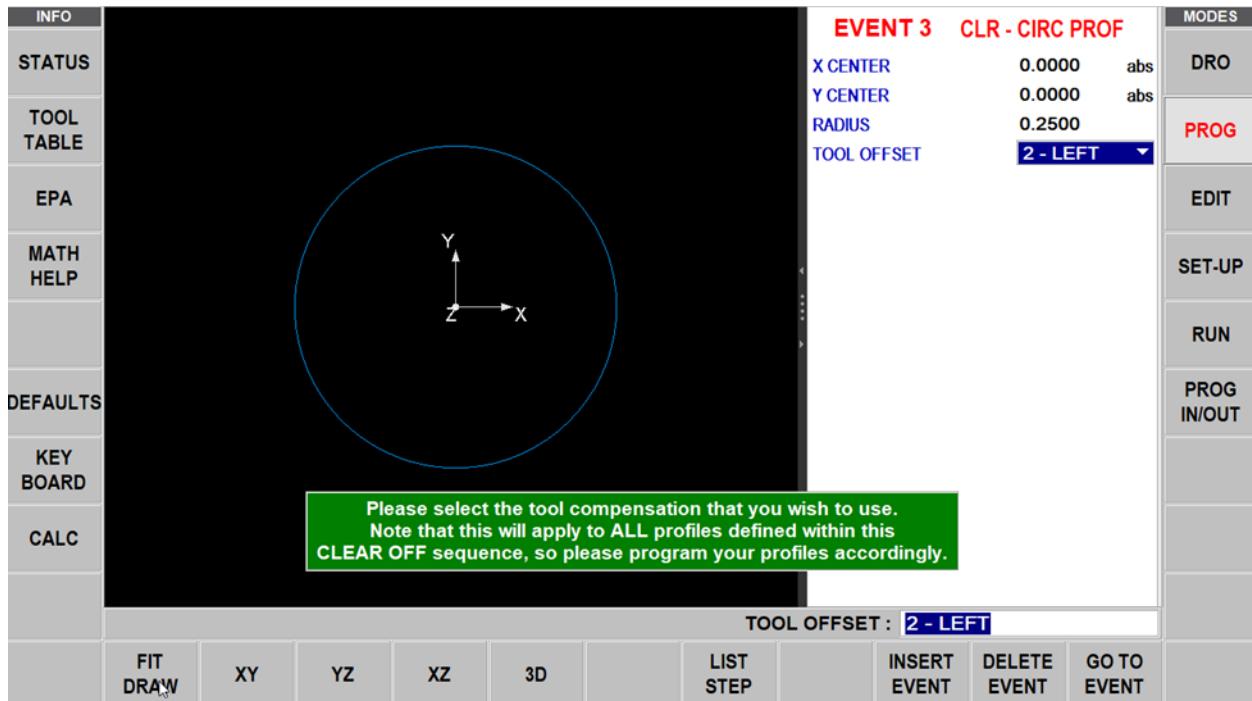
Figure 9.2.17b Tool Offsets should be same for all profiles inside a Clear Off event

Once all profiles are created, Click END PROFILES
Once Profiles are defined, you will have a choice of stock shapes:
CIRCLE STOCK – The circumference defined.
RECTANGULAR STOCK – A rectangular shape with all corners 90-degree right angles and the sides parallel to the X and Y axes.
IRREGULAR STOCK – Initiates the powerful Auto Geometry Engine for you to define a shape made up of straight line and arcs.
Clear Off can be used with both Parasolid and DXF options.

9.3   Finishing Teach Events
Teach events are either POSN, DRILL or MILL events that are originated in the DRO Mode (see Section 7.10).

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.
- **TEACH DRILL:** This may also be made into a bore or tap event.
- **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.

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.

10.0 Program Mode Part 3: Irregular Profiles, Pockets and Islands

The powerful Auto Geometry Engine® (A.G.E.)
When you program an Irregular Pocket or an Irregular Profile the Auto Geometry Engine® (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 RMX CNC. 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.

10.1 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. Complete this first event as you would for any mill event except there is no prompt for X or Y end.

The second event that defines the end point of the first line or arc in the profile must have a known X end and Y end (and X Center and Y Center) dimensions. If that is not available on the print, you may be able to enter a ramp on line to the first point. The beginning of this first line would be the point entered in the header (first) event.

INFO		EVENT 1	IRR PROFILE		EVENT 2	A.G.E. MILL	OK	MODES
STATUS		X BEGIN	0.0000	abs	TANGENT			DRO
		Y BEGIN	0.0000	abs	X END	2.0000	abs	
EPA		Z RAPID	0.1000	abs	Y END	0.0000	abs	PROG
		Z END	-0.5000	abs	CONRAD	0.0000		
TOOL TABLE		TOOL OFFSET	RIGHT ▼		LENGTH			EDIT
		# PASSES	2		LINE ANGLE			
KEY BOARD		FIN CUT	0.0100					SET-UP
		RPM	2000.00					
MATH HELP		FIN RPM	2500.00					RUN
		Z FEED PER MIN	10.0					
		XYZ FEED PER MIN	20.0					PROG IN/OUT
		FIN FEED PER MIN	15.0					
		TOOL #	1					
CALC		FIN TOOL #	2					
DEFAULTS								
UNDO								

	FIT DRAW	XY	YZ	XZ	3D		LIST STEP		INSERT EVENT	DELETE EVENT	GO TO EVENT

Figure 10.1 - Once the header event is complete, you will choose between A.G.E. Mills or A.G.E. Arcs.

With the header event complete, you'll define the pocket or profile with a series of Mills and Arcs:

- 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.

10.2   Prompts in A.G.E. Arc Programming

TANGENT: This refers to the tangency of the mill to the previous event (Section 10.10).

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

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

CONRAD: The dimension of a tangential radius to the next event.

LENGTH: The length of the mill from beginning to end.

LINE ANGLE: The angle of this mill line (moving from begin to end) measured counterclockwise from the positive X axis (that is 3 o'clock) (Section 10.11).

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 (Section 10.8).

10.3   Prompts in A.G.E. Arc Programming

TANGENT: This refers to the tangency of the mill to the previous event (Section 10.10).

DIRECTION: The clockwise or counterclockwise direction of the arc.

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

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

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

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

CONRAD: The dimension of a tangential radius to the next event.

RADIUS: The radius of the arc.

CHORD ANGLE: The angle spanned by the arc (Section 10.12).

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 (Section 10.8).

10.4   Skipping Over Prompts

In the Auto Geometry Engine®, events don't have to be complete before you can go to the next one. You can skip the data you don't know by tapping the next prompt. You may also simply swipe the event to the left and get started on the next one.

You should always select whether or not the next event is tangent to the previous event (Section 10.10).

10.5   OK / NOT OK

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. Swipe to the next event.

When the A.G.E. has sufficient data, it will calculate the data needed to fully define the previously NOT OK events. All data calculated by the A.G.E. is shown in red.

10.6   Ending A.G.E.

Any time all the events are 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 of the first event so that the pocket is closed. Otherwise, the ProtoTRAK RMX 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.
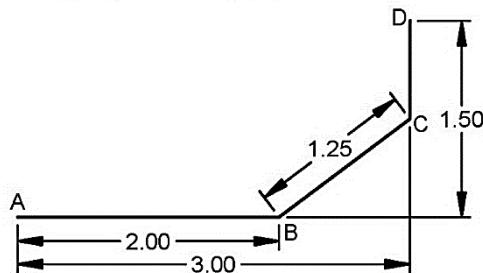
10.7   Reopening A.G.E.

To reopen the A.G.E. Profile or Pocket, simply use the BACK hard key or swipe to get one of the A.G.E. events on the right side of the screen. You may insert other A.G.E. Mill and A.G.E. Arc events.
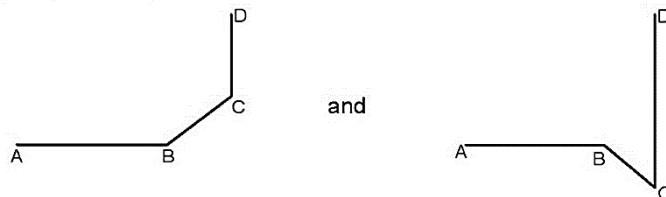
10.8   Guessing the Ends and Centers

Whenever you are missing X or Y Ends or Centers, you should generally enter a guess. That is because when the AGE computes X and Y positions it occasionally finds more than one solution. The purpose of guessing is to help AGE decide the correct solution.

We want to machine from A to B to C to D. The length of line BC is known but not the Y dimension of point C.

When this information is given to AGE without guessing two solutions can be calculated.

and

Guessing the Y location of C allows AGE to calculate the correct postition.

Figure 10.8 - Example of A.G.E. computing X & Y solutions without using guess.

Guess by selecting the GUESS soft key and then tapping the screen drawing approximately where the end or center point is. Guesses usually don't have to be very accurate unless the part is small and intricate and has multiple solutions close to each other.

Note - *When you guess the X End or X Center, the A.G.E. will also input a guess for the Y dimension. If the Y End or Y Center dimensions are known, go back and correct the guessed data so that the A.G.E. has more data to work with.*

10.9   Calculated Data

Prompts that are skipped or for which guesses are entered may be replaced by data calculated by the ProtoTRAK RMX CNC. 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 tap the prompt and re-input the data.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

10.10   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 and would not cross each other if extended. 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.
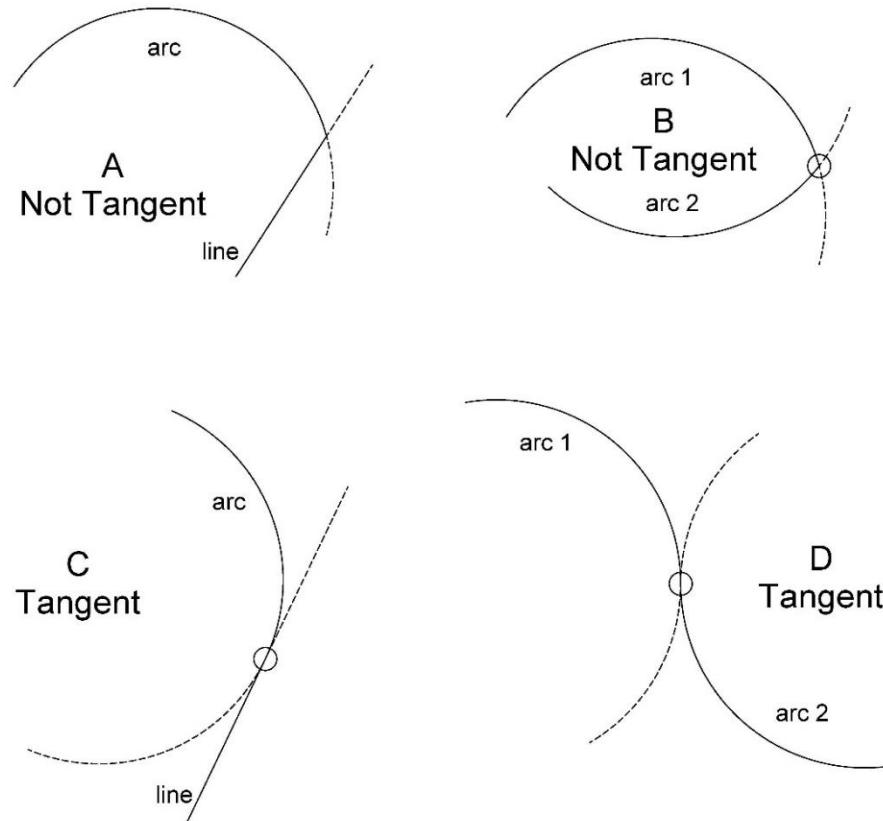
Figure 10.10 - Events are tangent when they share one and only one point and would not cross each other if extended.

Refer to Figure 10.10

- A line and an arc or 2 arcs are not tangent if you extend the lines past where they meet and they cross each other. A and B are not tangent.

- A line and an arc or 2 arcs are tangent if you extend the lines past where they meet and they do not cross each other. C and D are tangent.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

10.11   Line Angle

The Line Angle prompt can be a powerful aid to the A.G.E. in helping you with incomplete print data.
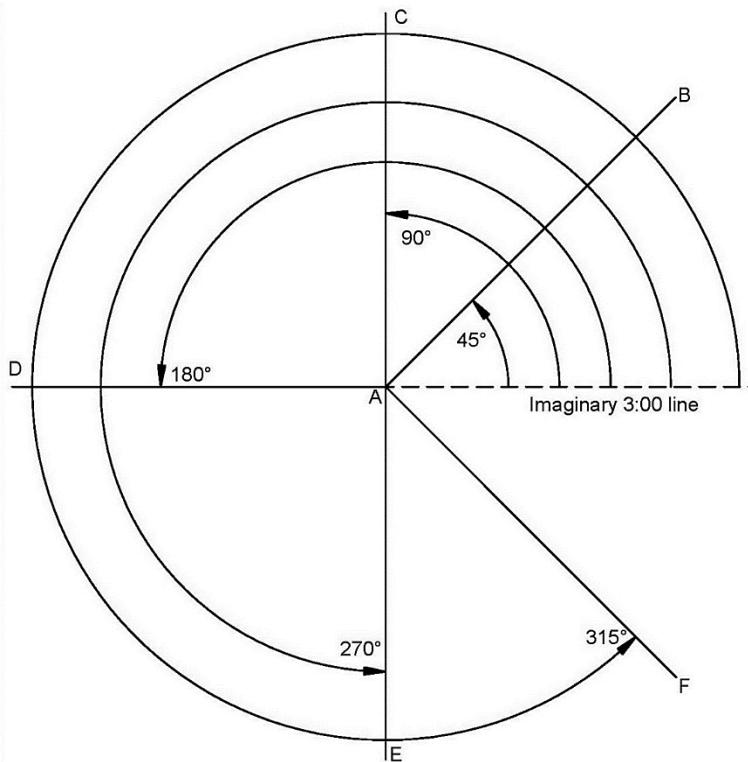
Figure 10.11a - Line Angles are referenced from an imaginary line at 3:00 o'clock.

To measure the angle of an angled line, imagine another line that starts at the beginning of the angled line and extends in the positive X direction or 3:00 o'clock. The line angle is the angle between the imaginary and angled lines always measured positive in the counter-clockwise direction.

In the drawing all lines to be machined from A to B or C or D or E or F and are measured from the imaginary dashed line that starts at A (the beginning) and goes in the positive X direction or 3:00 o'clock. The line angles are shown in the drawing.
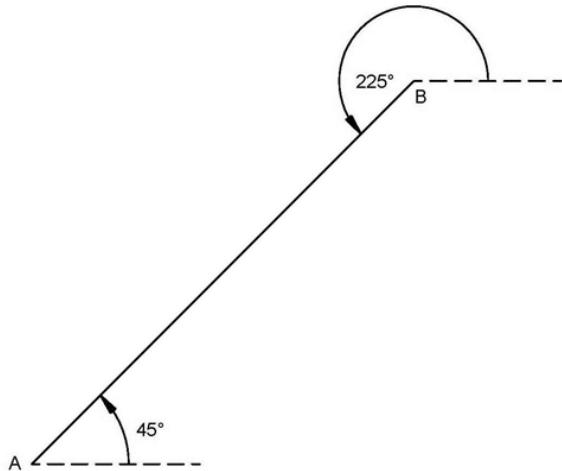
Figure 10.11b - The imaginary line is drawn from the beginning of the angled line to be machined.

The imaginary positive X or 3:00 o'clock line must always be drawn from the beginning of the angled line to be machined. Referring to Figure 10.10b above, when machining from A to B, the angle of the line is 45° measured from the imaginary line at A. But when machining from B to A, the angle of the same line is 225° (180° + 45°) because B is the starting point and the imaginary line must be drawn from there.

10.12   Chord Angle
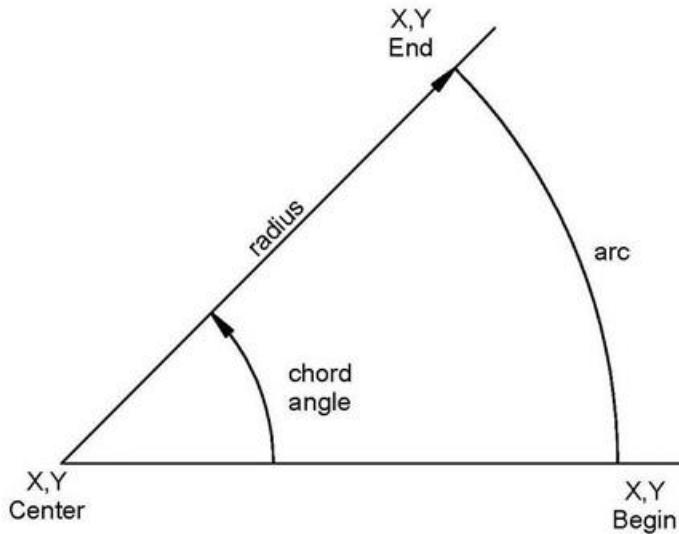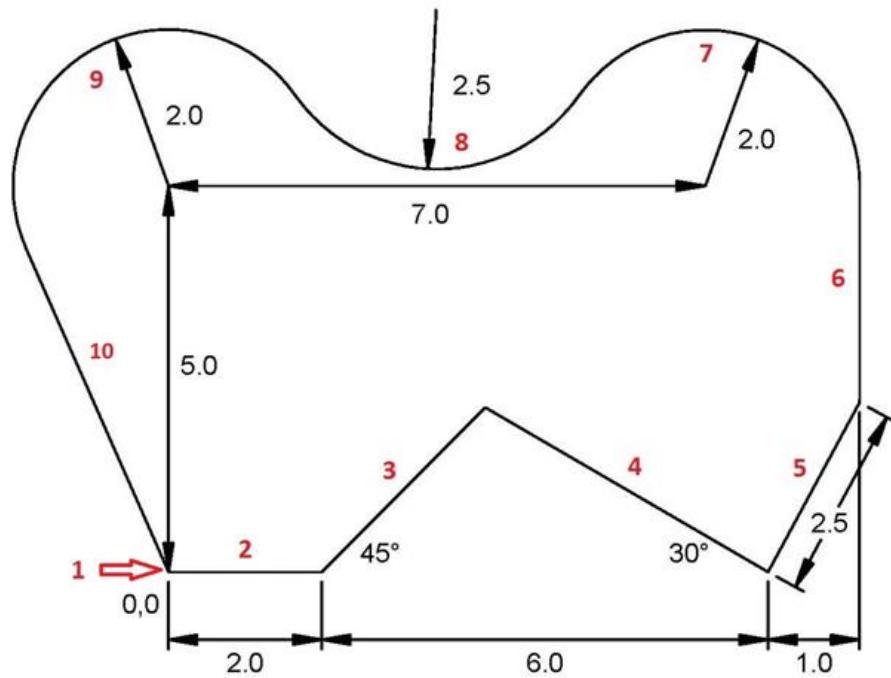
Figure 10.12 - The chord angle is measured counter-clockwise.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

10.13   Sample A.G.E. Program

Consider this profile made up of 9 lines and arcs
Notice what is not known:

- Intersection of lines 3 and 4
- Intersection of lines 5 and 6
- Intersection of arcs 7 and 8
- Center of arc 8
- Intersection of arcs 8 and 9
- Intersection of arc 9 and line 10

Event 1

The first programmed event is known as the Header. In this sample, this will be Event 1. To begin the A.G.E. you must start by programming the Header and a mill or arc that is fully known, that is, all the needed dimensions are there. The 2" line segment (2) starting at 0,0 is fully known, so this is where we'll start.

Complete this first or header event as you would any mill event except there is no prompt for X end or Y end.

Event 2

Select AGE Mill

- Input X end (2.0) and Y end (0) as shown on the screen
- There is no CONRAD to the next event
- Note the OK label next to the event line

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
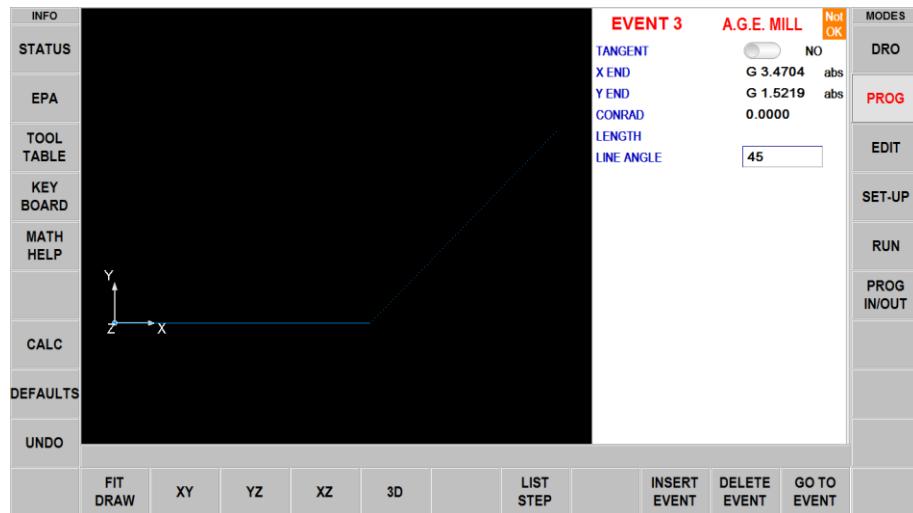
Event 3

Select AGE MILL

The line in event 3 is not tangent to the line in event 2 (no 2 straight lines can be tangent to each other) X end and Y end are not known. Tap Guess and select the screen above and to the right of Line 1 for each dimension.

Line angle is 45° measured from an imaginary positive X line from the beginning of Line 2
The line length is not known, skip.

Note the "NOT OK" label. We have input everything we know so swipe to the next event.

Event 4

Select AGE MILL

- The line in event 4 is not tangent to the line in event 3.
- X end is known (8.0) and Y end is known (0).
- The line angle is 330° measured from an imaginary position X line from the beginning of Line 3 in a counter-clockwise direction.

Note - *This event as well as the previous event are now OK.*


Event 5

Select AGE MILL
- X end for line 5 is known (9.0) but Y end is not known. Tap Guess and tap the screen approximately where the Y end is.
- The 2.5 length is known.
- Note the event is OK.

INFO		EVENT 5	A.G.E. MILL	OK	MODES
STATUS		TANGENT	NO		DRO
EPA		X END	9.0000 abs		PROG
		Y END	G 2.5415 abs		
TOOL TABLE		CONRAD	0.0000		EDIT
		LENGTH	2.5000		
KEY BOARD		LINE ANGLE			SET-UP
MATH HELP					RUN
					PROG IN/OUT
CALC					
DEFAULTS					
UNDO					
	FIT DRAW	XY	YZ	XZ	3D		LIST STEP		INSERT EVENT	DELETE EVENT	GO TO EVENT

Event 6
Select AGE MILL
- The line in event 6 is not tangent to the line in event 5.
- X end (9.0) is known and the 5.0 Y end is known.
- Note the event is OK.

INFO		EVENT 6	A.G.E. MILL	OK	MODES
STATUS		TANGENT	NO		DRO
EPA		X END	9.0000 abs		PROG
		Y END	5.0000 abs		
TOOL TABLE		CONRAD	0.0000		EDIT
		LENGTH			
KEY BOARD		LINE ANGLE			SET-UP
MATH HELP					RUN
					PROG IN/OUT
CALC					
DEFAULTS					
UNDO					
	FIT DRAW	XY	YZ	XZ	3D		LIST STEP		INSERT EVENT	DELETE EVENT	GO TO EVENT

Event 7

Select AGE ARC

- The arc is tangent to the line in event 6.
- The arc is counter-clockwise
- X end and Y end are not known. Tap Guess and tap the screen approximately where Arcs 6 and 7 meet.
- X center at 7.0 and Y center at 5.0 are known
- The radius at 2.0 is known

This is all we know. Note the "NOT OK" label.
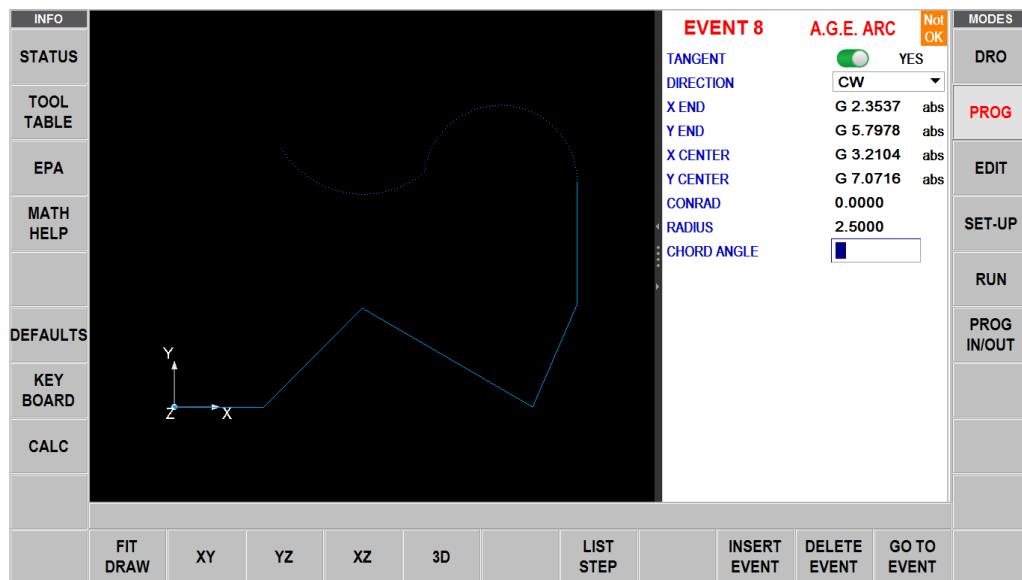
Event 8

Select AGE ARC
- The arc is tangent to Arc 7.
- The arc is machined clockwise.
- X end and Y end are not known. Tap Guess and tap the screen approximately where Arcs 8 and 9 meet.
- X center and Y center are not known. Tap Guess and tap the screen approximately at the center of Arc 8.
- The radius is 2.5.

This is all we know.

Note - *Event 8 is NOT OK and Event 7 is NOT OK yet either. No worry, page forward to next event.*

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
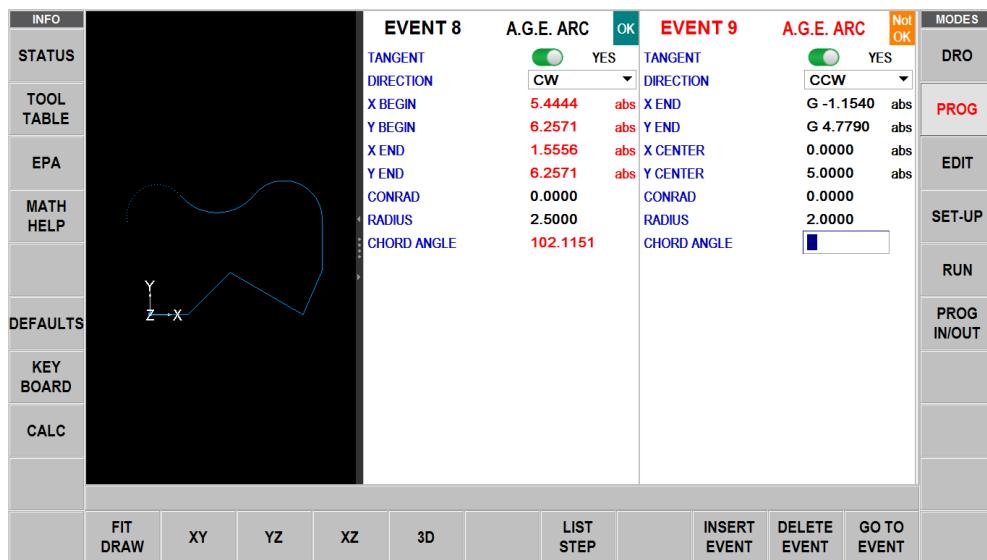
Event 9
Select AGE ARC
- The arc is tangent to Arc 8
- The arc is machined counter-clockwise
- X end and Y end are not known. Tap Guess and tap approximately where Arc 9 and Line 10 meet
- The arc center is known: X center 0 ABS and Y center 5 ABS
- The radius is 2.0

Note - *Event 9 is NOT OK but now Events 7 and 8 can be fully calculated and are OK. Page forward to next event.*
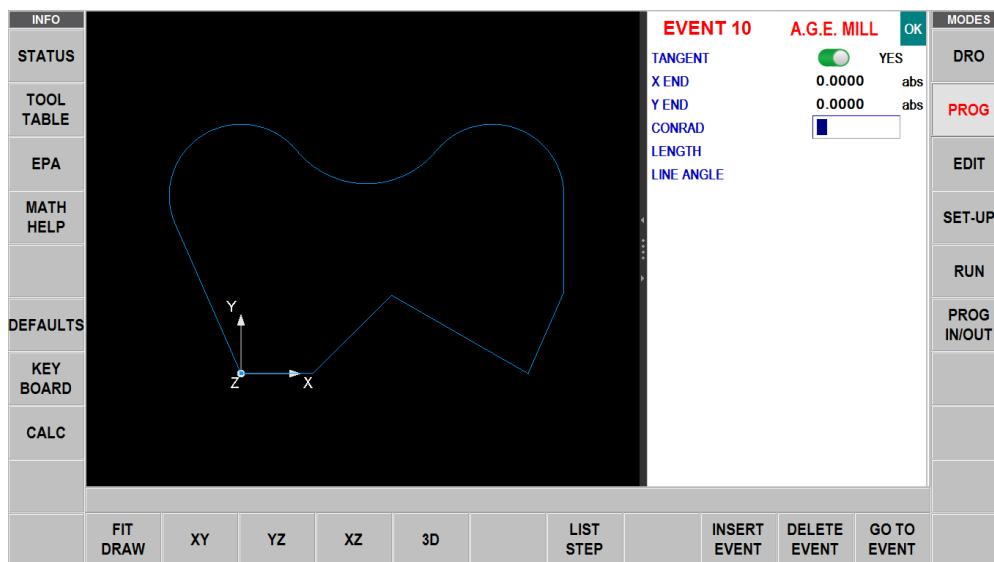
Event 10
Select AGE MILL
- Line 10 is tangent to Arc 9
- X end at 0.0 and Y end at 0.0 are known

Note - *Event 10 is OK as well as Event 9. The entire profile has now been programmed.*

- Page forward and tap END AGE

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

11.0 Edit Mode

Within Program Mode, you can recall and re-input data event by event. In contrast, 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, you must store the program again under the same name in the In/Out Mode.

11.1   Delete Events

Rather than delete one event at a time in Program Mode, you may delete a group of events in the Edit Mode.

In Edit Mode, press Delete Events.

The Data Input Line will prompt for the first event and then the last event to be deleted. Input the event numbers and press SET. The remaining events will be renumbered.

11.2   Search Edit

Selecting Search Edit opens the powerful Spreadsheet Editing function of the ProtoTRAK RMX. Spreadsheet Editing allows you to view program data in a table. From there you can make global changes to the program by selecting which event prompts you want to view, sort or edit. This is particularly useful if you are working with a large program and you need to make changes to many events.

When you press the SEARCH EDIT soft key, the screen will load a table that contains data for the entire program sorted by event number. The first view will show events 1 – 17. Use the Page and Data soft keys, swipe the screen or tap the scroll bar to see other events.

INFO	EV # ▲	TYPE	TOOL #	RPM/SS	FIN RPM/SS	Z FEED	XYZ FEED	FIN FEED	Z RAPID	Z END	MODES
STATUS	1	MILL	1	2000.00 R		5.0	20.0		0.1000 abs	0.0000 inc	DRO
	2	MILL	1	2000.00 R		5.0	20.0		0.0000 inc	0.0000 inc	
TOOL TABLE	3	MILL	1	2000.00 R		5.0	20.0		0.0000 inc	0.0000 inc	PROG
	4	MILL	1	2000.00 R		5.0	20.0		0.0000 inc	0.0000 inc	
EPA	5	DRILL	2	3000.00 R		7.5			0.1000 abs	-0.6000 abs	EDIT
	6	DRILL	2	3000.00 R		7.5			0.0000 inc	0.0000 inc	
MATH HELP	7	DRILL	2	3000.00 R		7.5			0.0000 inc	0.0000 inc	SET-UP
	8	DRILL	2	3000.00 R		7.5			0.0000 inc	0.0000 inc	
OPTIONS	9	DRILL	2	3000.00 R		7.5			0.0000 inc	0.0000 inc	RUN
	10	DRILL	2	3000.00 R		7.5			0.0000 inc	0.0000 inc	
DEFAULTS	11	DRILL	2	3000.00 R		7.5			0.0000 inc	0.0000 inc	PROG IN/OUT
	12	DRILL	2	3000.00 R		7.5			0.0000 inc	0.0000 inc	
KEY BOARD	13	CIRC PCKT	3	2500.00 R		7.5	20.0		0.1000 abs	-0.2000 abs	
	14	CIRC PCKT	3	2500.00 R		7.5	20.0		0.0000 inc	0.0000 inc	
CALC											

TOOL # :   1

| PAGE FWD | PAGE BACK | DATA TOP | DATA BOTTOM | GO TO EVENT | CHANGE ALL | | | | RETURN |

Figure 11.2 - The Search Edit key launches powerful Spreadsheet Editing.

Each row in the spreadsheet represents the data for the event number shown in the first column on the left. The event number and event type are always displayed in the first two columns, 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.
- **DATA TOP:** Highlights the data in the first row of the current sort.
- **DATA BOTTOM:** Highlights the data in the last row of the current sort.
- **GO TO EVENT:** Highlights the data in the row of the corresponding event.
- **CHANGE ALL:** Enables you to make global changes of data, described below.

Note - *The EV# (event number) and (event) TYPE may not be edited in Search Edit so the highlighter will not go there.*

11.2.1 Selecting Data to be Displayed on the Search Edit Table

In order to change the data selected in the table, tap the Options info key. You can choose which data to display and the order of the columns by opening or closing the slide bar. Tap Reset to clear all the selections. Tap Options to close and the newly selected prompts will show in the table.

Figure 11.2.1 - The Options Info key in search edit enables you to pick the fields to display.

11.2.2 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.

Simply tap the heading of the column to sort its data in ascending order (smallest first) and tap again to change to descending order. The secondary sort will always be Event #.

11.2.3 Making Changes to Data

To edit a single entry, tap the entry, input the change and press INC SET or ABS SET.

To edit a group of identical entries:
1. Sort on the entry prompt by tapping the column head. You may sort on any value that gets the data you want to change together.
2. Tap on the first data value you want to change (the highest on the list).
3. Tap Change All.
4. All data values that are the same and below the one you tapped will be highlighted.

140

5. Input the new value and ABS SET or INC SET. All highlighted entries will be changed to the new value.

The sequence below illustrates changing the # pecks in the program.

Figure 11.2.3a - First, the data is sorted by Event in ascending order to get the drill events together and then # pecks in event 5 is selected.

Figure 11.2.3b - Next, tapping the Change All soft key highlight all the same # pecks from event 5 and down.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

INFO	EV #▲	TYPE	TOOL #	RPM/SS	Z FEED	XYZ FEED	FIN FEED	Z RAPID	Z END	# PECKS		MODES
STATUS	1	MILL	1	2000.00 R	5.0	20.0		0.1000 abs	0.0000 inc			**DRO**
	2	MILL	1	2000.00 R	5.0	20.0		0.0000 inc	0.0000 inc			
TOOL TABLE	3	MILL	1	2000.00 R	5.0	20.0		0.0000 inc	0.0000 inc			**PROG**
	4	MILL	1	2000.00 R	5.0	20.0		0.0000 inc	0.0000 inc			
EPA	5	DRILL	2	3000.00 R	7.5			0.1000 abs	-0.6000 abs	2		**EDIT**
	6	DRILL	2	3000.00 R	7.5			0.0000 inc	0.0000 inc	2		
MATH HELP	7	DRILL	2	3000.00 R	7.5			0.0000 inc	0.0000 inc	2		**SET-UP**
	8	DRILL	2	3000.00 R	7.5			0.0000 inc	0.0000 inc	2		
OPTIONS	9	DRILL	2	3000.00 R	7.5			0.0000 inc	0.0000 inc	2		**RUN**
	10	DRILL	2	3000.00 R	7.5			0.0000 inc	0.0000 inc	2		
DEFAULTS	11	DRILL	2	3000.00 R	7.5			0.0000 inc	0.0000 inc	2		**PROG IN/OUT**
	12	DRILL	2	3000.00 R	7.5			0.0000 inc	0.0000 inc	2		
KEY BOARD	13	CIRC PCKT	3	2500.00 R	5.0	20.0	0.0	0.1000 abs	-0.2000 abs			
	14	CIRC PCKT	3	2500.00 R	5.0	20.0	0.0	0.0000 inc	0.0000 inc			
CALC												

OF VARIABLE PECKS : 2

PAGE FWD	PAGE BACK	DATA TOP	DATA BOTTOM	GO TO EVENT	CHANGE ALL				RETURN

Figure 11.2.3c - Finally, the # pecks was changed to 2 in all the events from event 5 and down by pressing 2, SET.

11.3   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 14.2.

11.4   G-Code Editor

The G-Code Editor is an Advanced Option which 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.

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

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. 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.

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.*

11.5   Update GCD

The UPDATE GCD button will show up when a .PTG or .GCD file is loaded into memory. This feature allows you to update the G code file without losing your pictures, notes, tool offsets etc. This is useful when you go back to your computer and need to generate a new G code file.

When a user selects Update GCD in EDIT mode, a PROG IN/OUT screen will open to allow the user to select a newer GCD file. Once the user presses UPDATE, the select GCD file will replace the current GCD file.

11.6   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 the all 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.

12.0 Set-Up Mode
12.1   Tool Path
When the TOOL PATH soft key is pressed, the program is processed and the tool path graphics are displayed. Large programs will take longer to process.

At the completion of running Tool Path, a time will appear in the upper left-hand corner of the screen with the estimated time to run the program. Since tool changes are manual, no time is allocated for the time required to change tools.

Most programming errors that would prevent the program from running are detected when the tool path graphics are selected. 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.
- Position and drill events are drawn in yellow.
- Rapid moves are in red.
- Programmed feeds are in blue.
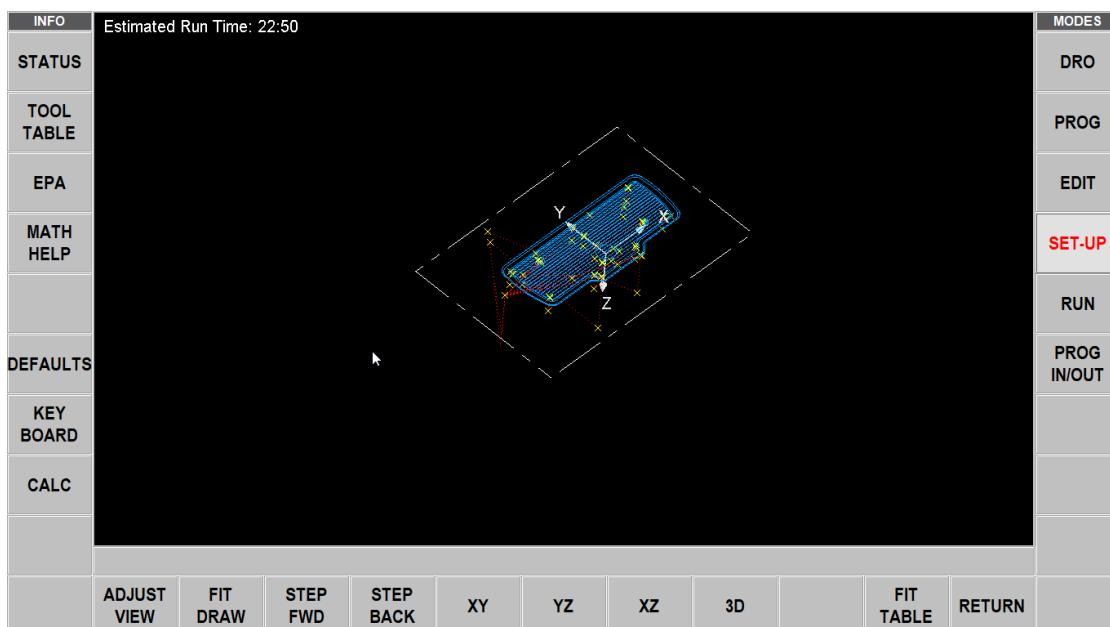
Figure 12.0 - The ProtoTRAK RMX processes the program and then displays the tool path.

When the drawing is displayed you may spread or pinch to zoom in or out. You may also pan up, down or sideways.

Soft Keys in Tool Path:
- **ADJUST VIEW:** Allows the user to move the drawing in all four directions and zoom in/zoom out.

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

- **STEP FWD:** Each press of the STEP FWD button shows the next tool move. To complete the drawing automatically, press FIT DRAW. Tap STEP to move through the path one move at a time. The move you are on will be shown in purple.

- **STEP BACK:** Each press of the STEP BACK button moves backward in the program 1 move.

    Note - *Each step forward or backward shows the X, Y and Z coordinate of that move.*

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

- **FIT TABLE:** Helps user verify that the drawing fits within the machine's table.

- **RETURN:** Returns to the previous soft keys, retaining the adjustments that were made to the drawing.

12.2 Fixture Offsets

Fixture offsets are entered in the Set-Up Mode.

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

Once you set your absolute zero on the base, it is a 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:
1. By entering the numbers with the keypad.
2. By positioning to the next fixture, tapping the offset value, and then pressing ABS SET.

The ERASE TABLE button is used to erase all fixture offsets from the table.

12.3 Verify Part

Verify Part is used to see a solid graphical representation of the tools making the part. There is no better way to see how your tool and part will interact than to see the solid model and tool represented in Verify. Of course, you must define the diameter of your tool along with your type of tool (finish end mill, ball end mill, drill, etc.) you are using when setting up your tools in tool setup. This allows you to draw the correct shape for the tools you are cutting with.

Press VERIFY PART and the following soft keys will show:
- **DEFINE STOCK:** If you wish to input dimensions of your rectangular stock relative to the programmed absolute zero.

    You don't necessarily have to define the stock because Verify will automatically calculate and define the stock based on the G code generated to run the program. The Z top and bottom value Verify calculates will be based off of the largest Z positive and negative value that is in the program. This means if you program a drill to go through the part it will make this value the Z bottom value.

    By using Define Stock you can put in the actual stock size and therefore get a more accurate picture of the interaction between the tool and the part.
    Note - *The stock size you end up with is saved with your program.*

- **AUTO:** Will have Verify recalculate the stock for you.

- **ENLARGE STOCK:** Each time you press this soft key, it increases stock by .1 inch

    Press MAKE PART and the display shows your stock material.

    The status line shows whether simulation is on or off, the speed of the verification, the status of STOP or GO and the tool number for the tool that is being simulated. The speed can be adjusted

from single stop all the way to a value of 20. The default speed is set to 1. Press the speed up or down buttons to adjust your speed. When on the single step speed, pressing GO will step the tool through each step of the program.

The first 4 soft keys are used to adjust the part view and can be tapped before the simulation or during the simulation when the tool is cutting the part.

VERIFY PART will launch the solid graphic part verification routine. A Yellow progress bar will appear as the simulation of the tool path takes place. After the tool path simulation finishes, the part will be drawn as it will appear after machining.

- **SHOW PART:** Will skip the tool path simulation and show the finished part.

- **SPEED UP and SPEED DOWN:** Adjusts the speed of the tool path simulation.

- **STOP:** Stops the tool path simulation

- **EXIT:** Exits the verification and returns you to the DEFINE STOCK page.

Verify in 2 Axis
When using VERIFY software in 2 axis mode on 3 axis products, Verify will set the stock size in Define Stock to 0 since there is no Z information in your program. By default, we will make the part 1" thick when showing the material. Manually enter Z information in Define Stock to get a good representation of the part.

12.4   Pictures and Notes
Pictures and Notes are useful features available in Setup mode that help the user include pictures of their setup with PART program so when they run this part at a later date, they can remember how to set up the fixture and part.

12.4.1   Pictures
To save pictures with the program, go to SETUP mode, select the PICTURES soft key and choose pictures that you want to add. You can add up to 10 pictures for any given program. Once the pictures have been added, you can view the pictures to any given part using the PREVIOUS and NEXT button. Once the last picture is viewed and you select NEXT, the first picture is displayed.

Figure 12.4.1 - Example of a saved picture.

12.4.2 Notes

The Notes function allows the user to include notes for themselves related to the program within the program for future reference. User has ability to have short and large notes. The software provides 4 pages for notes in which the first page has 9 short boxes for notes plus 1 large box. The following pages have 1 large box each for notes.

Figure 12.4.2 - Example of the Notes Function screen.

12.5   Run Strategy

Run Strategy is a soft key which allows the user to select the run strategy when making the same part multiple times using fixtures. The Default Run Strategy is Run part by part. The user can select either Tool by Tool or Part by Part.

The selected strategy does not get saved within the program or during reboot.
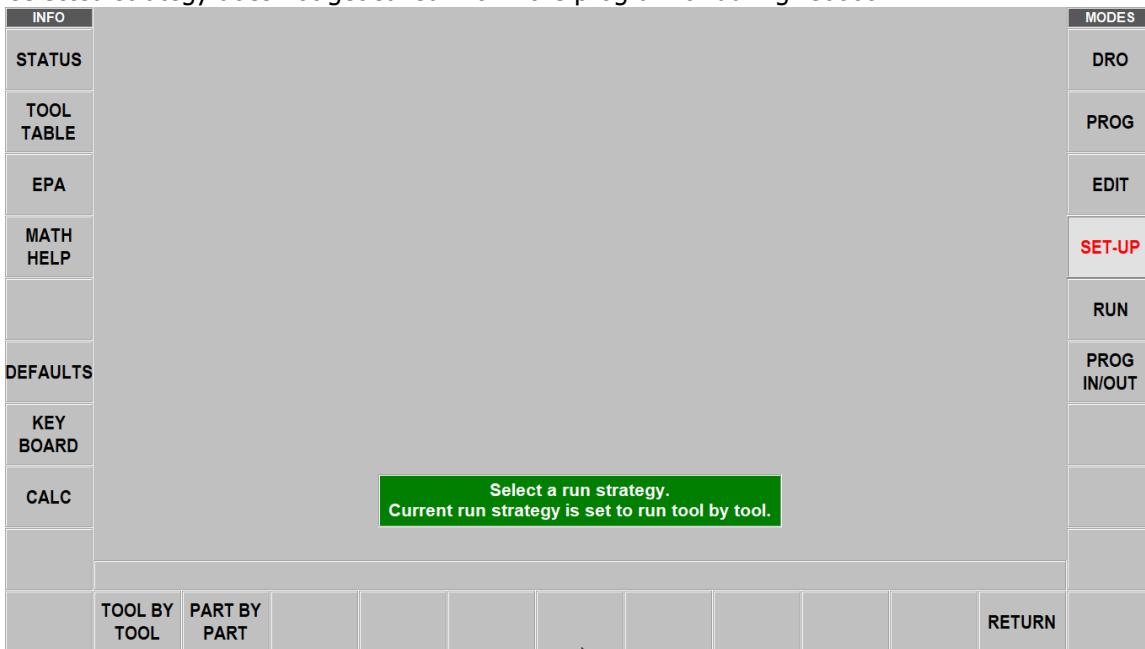
Figure 12.5 - ProtoTRAK RMX displaying run strategy soft keys with Tool by Tool run strategy set.

12.6   Z Retract

Note - *Z Retract is only available on VMC2.*

Z Retract is a parameter that provides the user with the ability to set the Z Retract where the tool goes, for the tool changes.

The value must be set in Set-Up mode in order for the current program to override the value set in Defaults. When set, the program will already have it saved, the next time the user pulls in the program.

If the value is not set in Set-Up mode, the Defaults' value will be used instead.

12.7   Service Codes

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

WARNING!
Before using service codes, be aware that some of the routines are very powerful and may change important system settings. Some of the routines cause the servos to come on and move at rapid speed.

The Service Codes are divided into logical categories. The table below summarizes the more important ones. See the service manual for more information on using Service Codes.

12.7.1   Software

148

Code	Description	Comment
33	Software, Firmware and PLC versions	Displays current software versions and system settings.
141	Load configuration file from USB thumb drive	To load configuration files from a USB thumb drive to the RX control.
142	Save configuration file to USB thumb drive	To save the configuration files for reloading later. When a computer replacement is necessary, saving the settings to a thumb drive for reloading them later is highly desirable.
316	Update Software	Runs the routine that copies new software from a USB thumb drive device to the ProtoTRAK system. Use this routine to install new ProtoTRAK software.
318	Converter and Software Options	Displays which software options are turned on. Options in bold letters mean the option is active.

12.7.2   Machine Set-Up

Code	Description	Comment
123	Calibration Mode	Use to calibrate the RMX control with a standard. **Note -** *When you have glass scale option, we calibrate both the motor and glass scale at the same time.*
128	Backlash Calibration Constant	Use to load backlash compensation for each axis.
134	Set Per-Axis Reversal Spike Reduction Parameter	Use to set a reversal spike reduction parameter for X, Y or Z and minimize reversal spikes on a ball bar so the overall circularity number is reduced.
308	Reverse EHW Direction	Change direction of Electronic Hand Wheel on X axis or Y axis
311	Run Machine with Spindle Off	Allows the user to run a program on the machine with the spindle off. This normally would be something a service technician might want to do more so than a user.
400	Update Foreign Language MLS File	Used to update the software language table. Please contact your distributor for available languages and for more information.
501	Z Tool Change Height Position	Used to set Z axis tool change position. This needs to be set only if factory defaults configuration was lost
505	Overtravel Soft Limits	Allow user to set soft limits for travel on all 3 axes.
510	Spindle Setup and Calibration	Allow for calibration of the spindle.
513	Reset ATC	Will allow user to reset ATC if the machine has faulted in the middle of a tool change operation, or if the user aborted in the middle of tool change
520	Set Tool Carrier Locations	Use to set the 8 ATC tool locations as well as the tool change height. (**NOTE -** *Available only on VMC2.*)
530	Serial Numbers	Service code to record pendant, computer module and machine serial numbers. A warning will be given to user until this requirement is met.
540	Disable File Based Write Filter	This service code must be used when changes need to make to the operating system. When networking the control or loading drivers, this service code must be run.
541	Turn CTS Option On or Off	Service code to turn the CTS option ON when it is purchased.

12.7.3   Diagnostic Codes

Code	Description	Comment
1	Program, Configuration File, Log Back up	The following service code captures all important data from the RX machine. This includes the user's program, configuration files, message log, fault log etc. This is meant to be used to help us solve problems that may pertain to our software. This file can then be emailed to our service department.
54	Continuous Run Mode	Cycles through the program in current memory.
81	Program Panel Keyboard Test	Gives a tone feedback to a button push and highlights the button.
131	Manual DRO	Turns off servos so you can check encoders. Shows motor encoders and glass scale counts and direction of counts.
132	Electronic Handwheel Test	Test the EHW signals
314	Toggle Test Indicator (Debug Lights)	Turns on diagnostic lights to help troubleshoot certain problems.
319	Message Log	Logs the machine as it runs. It captures all key presses, error messages and data as the machine runs. It is saved automatically with service code 1.
320	Fault Log	A log that captures all faults and error messages.
326	Look Up Message	Allows user to display safety flashing messages, master messages and slave error messages on screen. Often used by translators converting English language to some other language.
327	Display Memory Check	Displays memory availability of various devices
521	Launch Hardware Tester	Allow to launch Hardware Tester UI on master and Hardware Tester 2 on the Slave
522	Diagnostic Charting Tool	Allow user to launch Diagnostic Charting Tool

12.7.4   Operator Defaults / Options
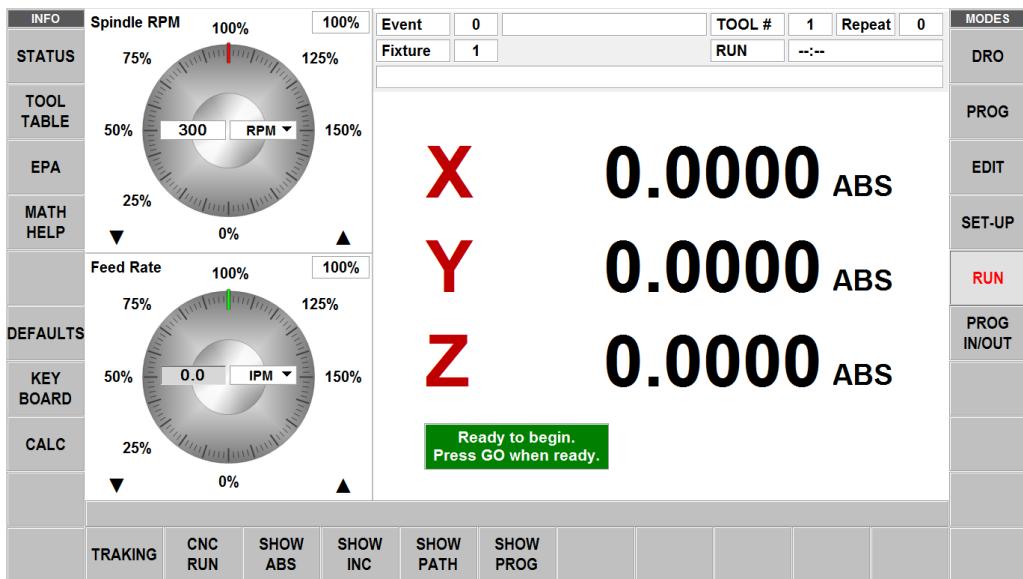
Code	Description	Comment
79	Turn On/Off Beeper	Turn the beeper on or off
328	EPA On/Off	Turns the EPA feature on or off
600	Manage User Profiles	Allows each user to save, delete, or reset individual default user profiles.

12.7.5   Lube Pump Codes

Code	Description	Comment
300	Lube Pump Switch Discharge	Allows the user to manual discharge the lube pump
301	Set Lube Pump Cycle Time	Sets the time in minutes between pumping cycles
302	Set Lube Pump Discharge Time	Sets the time in seconds for how long to pump for each cycle.

13.0 Run Mode

13.1 Run Mode Screen

Items on the Run Screen:

Status Bar: The Status Bar in Run mode displays 16 field boxes on the VMC2 and 17 field boxes on the TMC's as shown in the figures below.

Figure 13.1a - VMC2 Status Bar displayed

Figure 13.1b – TMC Status Bar displayed for Bolt Hole event with ATC Tool

- **Status Bar Box #1:** Event Label
- **Status Bar Box #2:** Event Value
- **Status Bar Box #3:** ATC Front **/#4:** ATC Back (*Available only on VMC2*)
- **Status Bar Box #3/#4:** ATC LOC label / ATC LOC # (*Available only on TMC's*)
- **Status Bar Box #5:** Tool loaded in spindle
- **Status Bar Box #6:** ATC LOC # of the tool in spindle
- **Status Bar Box #7:** Tool Diameter
- **Status Bar Box #8:** Tool Type
- **Status Bar Box #9:** Fixture Label
- **Status Bar Box #10:** Fixture Value
- **Status Bar Box #11:** Repeat Label

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

- **Status Bar Box #12:** Repeat Value
- **Status Bar Box #13:** Placeholder
- **Status Bar Box #14:** Run Label
- **Status Bar Box #15:** Total Run Time
- **Status Bar Box #16:** Event Comments
- **Status Bar Box #17:** Event Name

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.

Tool #: The current tool number to load when prompted.

Fixture: The fixture where machining will begin.

Run: Shows RUN or TRAKing when in TRAKing mode. The box to the right shows the runtime clock. This does not count the time for a manual tool change. You must first run tool path in setup mode.

13.2   Starting to Run

Before running a part, you must establish the position relationship between the part and spindle. That is, you need to identify where the part is on the table relative to the tool or spindle 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 spindle 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 two ways:
1. **START:** 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.

2. **START EVENT #:** Allows you to start in the middle of a program. When appropriate, you will be prompted to identify the repeat number you wish to begin or select between beginning at the Rough, Rest or Finish Tool. In addition to this, you can start at a certain pass # or at the XY, Z or XY and Z finish cut.

13.3   Program Run

SHOW ABS: The absolute X, Y, Z positions as the part is run.

SHOW INC: The incremental (or distance to go within the event) X, Y, Z positions as the part is run.

SHOW PATH: The tool path graphics as the part is run.

SHOW PROG: 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 in the green message box and proceed by pressing the GO key.

At the beginning of a program or once the STOP hard key is pressed, additional softkeys will be available:

- **TRAKing:** Tap this softkey to control the X, Y and Z programmed motion with one of the electronic handwheels. The TRAKing option must be active for this function.

- **CNC Run:** Tap this softkey to start the CNC run.
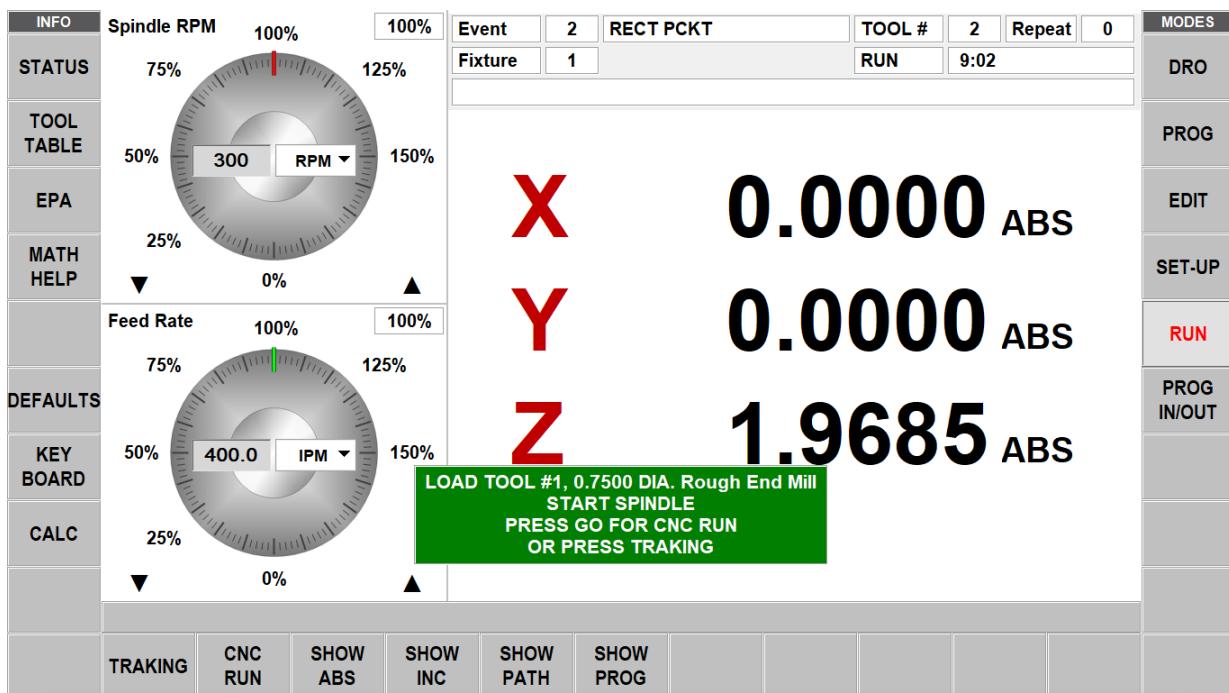
Figure 13.3 - ProtoTRAK RMX Displaying Programmed Run

13.4   TRAKing

TRAKing is a special kind of CNC run. When you tap the TRAKing soft key, the programmed head, table and saddle motion are controlled by using the Front Panel Electronic Handwheel or the optional electronic handwheels.

For the Front Panel Electronic Handwheel, moving the knob in clockwise direction moves forward through the program; moving counterclockwise moves backward through the program. The speed at which you move is decided by the pressing .0001"/.002 mm, .0010"/.020mm, .0100"/.200mm, and fast speeds.

For the optional electronic handwheels, 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.

13.4.1 TRAKing in Two-Axis CNC

When running the ProtoTRAK TMC 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.

Note - *TRAKing is not available in Two-Axis mode on the VMC2.*

13.5 Program Run Messages

While in the Run Mode, clear instructions and prompts from the RMX 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 in the box directly above the XYZ Dimension readouts.

Once the program starts, a Run Time Clock will appear near 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.*

13.6 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 tapping the CNC RUN softkey or pressing the **GO** key. You may also run the program by using the Electronic Handwheels on the table or saddle by tapping the **TRAKing** softkey.

13.7 Chip Clear

Whether or not you are using CNC Run or the (optional) TRAKing, you will have the Chip Clear option available once the running commences. Chip Clear is a handy way to interrupt the cutting and move the tool up and out of the part. You may also move the X & Y axis to a desirable location by using jog or turning the handwheels. You may stop the spindle if desired. When you are ready, tap RESUME and the tool will move back where it left off when you tapped CHIP CLEAR. X&Y will move to the previous location first, and then Z will come down to where you left off. When you hit the RESUME button, you can use CNC RUN or TRAKing to move back down into the part.

13.8 Feedrate and Speed Overrides

In program Run Mode, the programmed XYZ axis feeds as well as the rapid speeds may be adjusted. Likewise, the programmed spindle speed may be adjusted.

See Sections 7.4 and 7.5 for using the Graphical Spindle and Feed Rate Override.

14.0 Program In/Out

The Program In/Out Mode is where you work with programs you want to store, or receive from others.
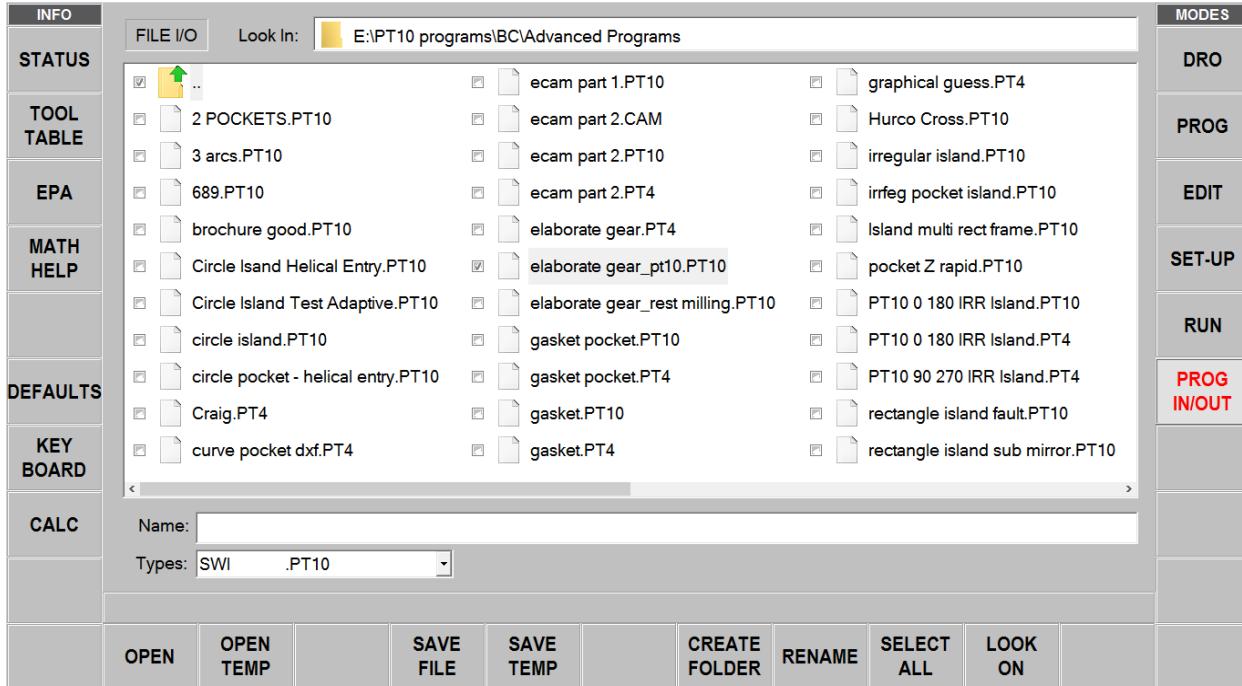
Figure 14.0 - The Program In/Out screen showing a few of the programs used to test the ProtoTRAK RMX before release.

14.1   Getting to Files and Directories
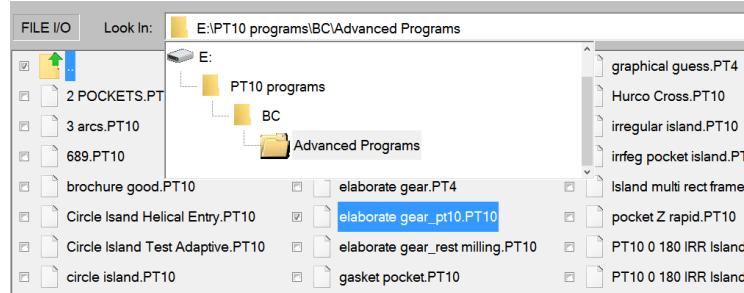
Figure 14.1 - Tap in the Look In box to see the available drives. Above, the file path is also shown for the highlighted program.

When you tap into the Look In box, the ProtoTRAK RMX will give you a view of the available drives. Tapping on the drive you want will expose its file structure in the main part of the screen. To open a drive, simply tap on the drive you want. Double tap the folder name or tap the Open soft key to open folders.

The Name box will display the name of the program in current memory until you highlight another folder or program.

The Types box will show which file types are displayed. If you purchased optional Converters, you will see the appropriate file type extension listed when you tap in the Types box. For example, if a DXF Converter is installed on the ProtoTRAK RMX, you will see the .DXF and .DWG extensions.

155

Thumb drives and Network locations are assigned to Drives D, E, F, etc. as you add them.

Note - Although you may view the contents, the ProtoTRAK RMX will not allow programs to be saved to the C Drive. You must use either a USB Thumb drive or Network to save and retrieve saved programs.

14.2 Saving a Program

To save a program you have written or edited, we recommend you do the following steps in order:
1. In PROG IN/OUT – open the folder in which you want to save the program.

2. The Program Name defined in the PROG mode will automatically be in the Name bar. When you press SAVE FILE, that file will be saved and should appear in the open folder.

3. Otherwise, you can click on the text box next to Name, and use the keyboard to type in your program name.

If you want to save a copy of the file under a different name, simply tap in the Name bar and the Key Board will automatically launch. Erase the old name and then enter the new name. Close the Key Board and tap SAVE FILE.

14.3 Opening a Saved Program

To open a program, navigate to the one you want and then tap on it to highlight. Then either double tap on the file name or tap the OPEN soft key.

The ProtoTRAK RMX will only keep one program in current memory at a time so you will be reminded that opening a new program will overwrite the one in current memory.

Tip - If you made changes to the program in current memory, save it again before opening another one.

When the new program is loaded into current memory, the ProtoTRAK display will revert to the startup screen.

The LOOK ON soft key will show you an XY line drawing of the program as you highlight them. If the program is large, the ProtoTRAK RMX may take a moment to display the drawing.

14.4 Temp Files

Temp files enable you to preserve the set-up information for the part you are working on, in addition to the program. This is handy in case you would like to shut down the ProtoTRAK RMX without losing the offsets, reference position and other information you need to run the program. You may also want to use Save Temp periodically as you are working if you are worried about losing power. It is three taps from programming to Save Temp and then back to where you were in the program.

All programs, tools and position info will be saved at the moment you tap Save Temp. Tapping Save Temp over-writes all data from previous Save Temp.
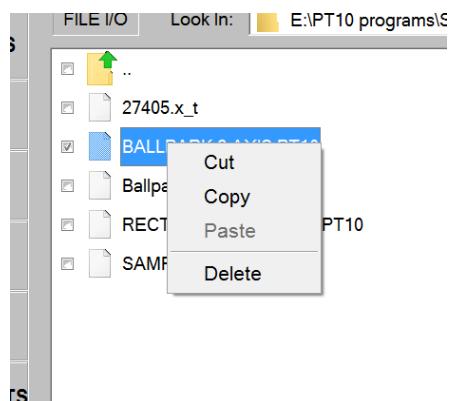
Tap Open Temp to restore all the information to the ProtoTRAK RMX.

Note - After Open Temp, it is wise to check X, Y, Z ABS zero in case something moved!

14.5 Deleting a Program or Folder

To delete a program or folder, press (not tap) and hold file name or folder in the data section until you see the highlighted circle, when you pick up your finger you will see the drop-down box. Press DELETE.

You will be warned before deleting.

Note - To delete the program from current memory, go to Edit Mode, Erase Prog.

14.6   Renaming a Program or Folder

To rename a program or folder from Program In/Out, first navigate to the item you would like to rename. Tap and highlight the Name box.

You will first need to erase the current name. Tap the Keyboard info key to launch the keyboard. Once it appears, tap into the Name box again so that what you do on the keyboard will be shown in the Name box.

Press Rename. The name of the file or folder will change. The old name will disappear (that is, you do not create a copy with a new name, leaving the old name as it was previously saved.)

14.7   Create a New Folder

To create a new folder, first highlight the drive or folder in which you want to have the new folder.

Launch the Keyboard by tapping the Keyboard info soft key. Then tap into the Name box. If there is a name already there, use the backspace of the keyboard to erase the name. The item with that name will not be affected.

Use the keyboard to write the name of the new folder. Press Create Folder and a new folder with the name in the Name box will appear.

14.8   Copy or Cut a File or Folder

To copy a file or folder, first navigate to the drive and double tap the folder where the program you want to copy is located.

Press (not tap) and hold file name or folder in the data section until you see the highlighted circle, when you pick up your finger you will see the drop-down box.
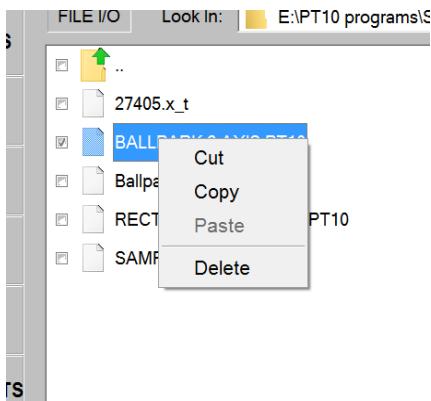
Figure 14.8 - Pressing and holding the file name will bring up the box as shown above.

Tap Copy to copy the item. Tap Cut to have the item deleted from the original location.

To place the copied item in a new location first navigate to the location where you want the copy to go. Press and hold this drive, folder name or screen within the folder as above and select Paste.

If you selected Copy originally, the file or folder will not be deleted from the first location. If you selected Cut, it will be deleted from the first location.

14.9 Backing Up Programs

You may do a backup using the procedure above. When you tap the Select All soft key, all the files and folders visible on the screen will be selected. By pressing Copy, you may place these files into a new location all at the same time.

Note - *If you select a large number of files, the ProtoTRAK RMX may need a moment to perform the copy and paste.*

14.10 Look On

Press the Look On button to enable a preview window on the left-hand side of the screen. This will dynamically show you a top view of the program that is highlighted on the right-hand side of the screen. Press the Look Off button to disable the preview window.

14.11   File Extensions

Programs you write on the ProtoTRAK RMX will automatically receive the file extension .PT10. The following is a list of other program types and what will happen if you open them.

Extension	Description	Notes
.PT10	ProtoTRAK RX series	RMX files created and files converted and then saved.
.PT4	ProtoTRAK S, V, SX, TX series	Some feature incompatibility.
.MX2	ProtoTRAK MX, AGE, M 2-axis series	Some feature incompatibility.
.MX3	ProtoTRAK MX, AGE, M 3-axis series	Some feature incompatibility.
.GCD	G-code Program	Will run as a G-code file.
.CAM	G-code	To be converted to ProtoTRAK format.
.DXF	A DXF file	Optional DXF converter will launch when the file is opened.
.DWG	An AutoCad DWG drawing file	Optional DXF converter will launch when the file is opened.
.PTG	Pictures & Notes feature w/ GCD files	Only available through Advanced Features with Verify.
.X_T	A Parasolid file	Optional Parasolid Converter will launch when the file is opened.
.MPF	A Siemens Sinumerik One (OUT) File	Will output MPF files to run in VMCsi machines

14.11.1   GCD vs PTG Files

1. GCD file will only contain the gcode and nothing more.

2. PTG file will contain and support the same gcode, but also tool info, pictures, notes, etc. Everything the user sets up in order to run his program.

3. When a PTG program is saved...ie "FileName.PTG," the following will also occur:
   - GCD \"FileName.PTG" folder will be created and the gcd file will be copied to this folder.
   - PICTURES\"FileName.PTG" folder will be create and included jpg files will be copied to this folder.

4. Cut, Copy, Paste will also include associated folder GCD and PICTURES and individual gcd and jpg files.

5. If a GCD or PTG program is opened fixture button will become work offset. Fixture table will show G54, G55, etc instead of fixture number.

6. Work Offset g codes in gcd program will be found and highlighted in red.

7. SAVE TEMP and OPEN TEMP will save and load GCD and PTG files.

8. When a GCD or PTG program is in memory, user can update only his gcode by going into EDIT and using UPDATE GCD feature. This will allow him to update the g code without losing his tool information, pictures, notes etc.

14.12   Supported G-Codes for CAM Converter

Open a G code file as a CAM file in order to convert it into an event program. Doing so will gain the advantage of being able to edit the program easier, such as using our Search Edit feature.

G Code	Description	Acceptable Format	Notes
G00	Position at Maximum Feedrate	G00 X0.0 Y0.0 Z0.0	
G01	Linear Interpolation	G01 X0.0 Y0.0 Z-1. F15.	F = Feedrate
G02	Circular Interpolation (Clockwise)	G02 X1. Y2. Z0. I0. J2. K0.	I, J, & K define the X, Y, & Z centerpoints respectively, incremental from the starting point of the arc.
G03	Circular Interpolation (Counter-clockwise)	G03 X2. Y0. Z0. I2. J0. K0.	
G06	Helix (Clockwise)	G06 X0. Y0. Z0. I1. J0. K0. D2.0	K defines the center point in Z of the first revolution of the helix.
G07	Helix (Counter-clockwise)	G07 X0. Y0. Z0. I1. J0. K0. D2.0	D defines the # of revolutions.
G20	Input in Inch	G20	
G21	Input in Metric	G21	
G40	Cutter compensation cancel	G40	
G41	Cutter compensation left	G41	
G42	Cutter compensation right	G42	
G54	Fixture offset 1 (Base offset)	G54	Note that this only calls out which fixture # is being used. Fixture Offsets (much like tool offsets) are defined conversationally at our control.
G55	Fixture offset 2	G55	
G56	Fixture offset 3	G56	
G57	Fixture offset 4	G57	
G58	Fixture offset 5	G58	
G59	Fixture offset 6	G59	
G73	Peck Drilling Cycle – Chip Break Peck	G73 X0. Y0. Z-1. R.1 Q.125 F15.	Q = depth of peck Chip break retract distance = 0.020"
G80	Canned Cycle Cancel	G80	
G81	Drilling Cycle	G81 X0. Y0. Z-1. R.1 F15.	Z = final depth R = rapid position
G82	Drilling Cycle, Dwell at Bottom	G82 X0. Y0. Z-1. R.1 P500 F15.	P = dwell time in milliseconds
G83	Peck Drilling	G83 X0. Y0. Z-1. R.1 Q.125 F15.	Q = depth of peck
G84	Tapping Cycle	G84 X0. Y0. Z-2. R.1 S300 F15.	S defines RPM Pitch = F / S
G85	Boring Cycle	G85 X0. Y0. Z-1 R.1 F15.	
G89	Boring Cycle, Dwell at Bottom	G89 X0. Y0. Z-1 R.1 P800 F15.	P = dwell time in milliseconds
G90	Absolute programming	G90	
G91	Incremental programming	G91	
M00	Pause / Program Stop	M00 (Set Z)	The comment within the parenthesis will be displayed in a green message box.
M05	Spindle Off	M05	
M06	Tool Change	T1 M06	
M07	Air/Mister On	M07	
M08	Coolant On	M08	
M09	Coolant Off	M09	
M21	Indexer Pulse - beginning of event	M21	
M38	Air/Mister Off	M38	

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

G Code	Description	Acceptable Format	Notes
M51	Auxiliary Output On	M51	
M53	Auxiliary Output Off	M53	
M88	CTS On	M88	
M89	CTS Off	M89	

14.13   Supported G-Codes for GCD Programs

Open a G code program as a GCD file in order to run the program as is. The advantage here is being able to run much larger files.

G Code	Description	Acceptable Format	Notes
G00	Position at maximum feedrate	G00 X0.0 Y0.0 Z0.0	
G01	Linear Interpolation	G01 X0.0 Y0.0 Z-1. F15.	F = Feedrate in Inches per minute
G02	Circular Interpolation (Clockwise)	G02 X1. Y2. Z0. I0. J2. K0.	I, J, & K define the X, Y, & Z centerpoints respectively, incremental from the starting point of the arc.
G03	Circular Interpolation (Counter-clockwise)	G03 X2. Y0. Z0. I2. J0. K0.	
G06	Helix (Clockwise)	G06 X0. Y0. Z0. I1. J0. K0. D2.0	K defines the center point in Z of the first revolution. D defines the # of revolutions.
G07	Helix (Counter-clockwise)	G07 X0. Y0. Z0. I1. J0. K0. D2.0	
G17	Select XY Plane	G17	G17 - G19 only apply to G2 and G3.
G18	Select XZ Plane	G18	
G19	Select YZ Plane	G19	
G20	Input in Inch	G20	
G21	Input in Metric	G21	
G40	Cutter compensation cancel	G40	
G41	Cutter compensation left	G41	
G42	Cutter compensation right	G42	
G54	Fixture offset 1 (Base Offset)	G54	The offset difference from each other is defined conversationally on our control.
G55	Fixture offset 2	G55	
G56	Fixture offset 3	G56	
G57	Fixture offset 4	G57	
G58	Fixture offset 5	G58	
G59	Fixture offset 6	G59	
G61	Exact Stop Movement	G61	Will feed into exact position at intersection points before proceeding to next event. May hesitate briefly.
G64	Blending Movement	G64	Will blend intersection points together for smoother motion.
G73	Peck Drilling Cycle – Chip Break Peck	G73 X0. Y0. Z-1. R.1 Q.125 F15.	Q = depth of peck
G80	Canned Cycle Cancel	G80	
G81	Drilling Cycle	G81 X0. Y0. Z-1. R.1 F15.	Z = final depth R = rapid position
G82	Drilling Cycle, Dwell at Bottom	G82 X0. Y0. Z-1. R.1 P500 F15.	P = dwell time in milliseconds
G83	Peck Drilling Cycle – Fixed Peck	G83 X0. Y0. Z-1. R.1 Q.125 F15.	Q = depth of peck

G Code	Description	Acceptable Format	Notes
G83	Peck Drilling Cycle – Variable Peck	G83 X0. Y0. Z-1. R.1 I.25 J.1 K.1 F15.	I = Initial peck amount J = Minimum peck K = Amount to reduce each peck by
G84	Tapping Cycle	G84 X0. Y0. Z-2. R.1 S300 Q.05	S defines RPM Q defines Pitch Feed = S x Q
G85	Boring Cycle	G85 X0. Y0. Z-1 R.1 F15.	
G89	Boring Cycle, Dwell at Bottom	G89 X0. Y0. Z-1 R.1 P800 F15.	P = dwell time in milliseconds
G90	Absolute Programming	G90	Default
G91	Incremental Programming	G91	
G94	Feed per Minute	G94	Default
G95	Feed per Rev	G95	
G98	Return Z to Initial Point	G98	Default
G99	Return to R point in canned cycle	G99	
M00	Program Stop (prompts for "Press Go")	M00	
M02	End of Program	M02	
M03	Spindle On - CW	M03	
M04	Spindle On - CCW	M04	
M05	Spindle Off	M05	
M06	Tool Change	T1 M06	
M07	Air/Mister On	M07	
M08	Coolant On	M08	
M09	Coolant Off	M09	
M21	Indexer pulse	M21	
M22	Move Table Towards Operator	M22	X will move to the location defined in DEFAULTS section. Y will move table all the way to front. Z will move to tool change.
M30	Program End	M30	
M38	Air/Mister Off	M38	
M51	Auxiliary Output On	M51	
M53	Auxiliary Output Off	M53	
M88	CTS On	M88	
M89	CTS Off	M89	
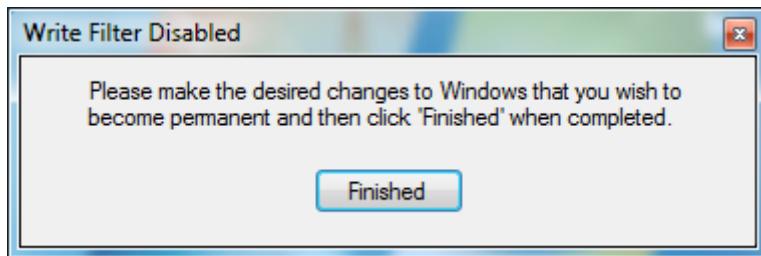
14.14   Networking

Prior to networking your control, you'll need to turn off the File Based Write Filter temporarily. In order to do this, you'll need to perform the following steps.

1. Press Mode, Setup, Service Code, input 540 and press ABS Set. When the following screen comes up, press Yes.

2. The machine will reboot and bring you to the Windows desktop. Here you'll make your changes, then once your changes are made, click Finished as shown below to reboot (Note, if you fail to click finish and reboot, you'll return to this screen. The RX software will not open up automatically on this screen)

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
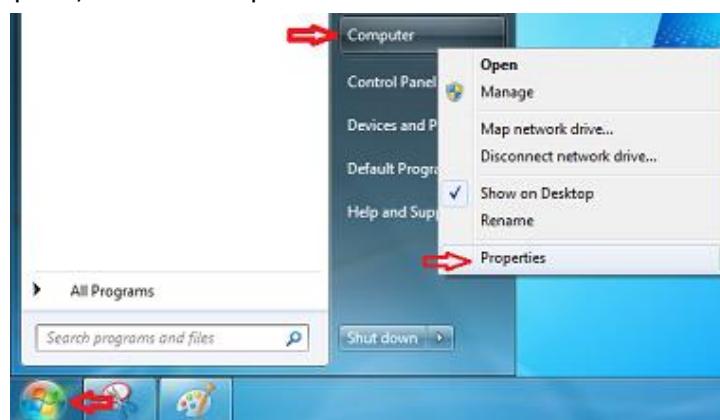
14.14.1 Networking – Windows 7
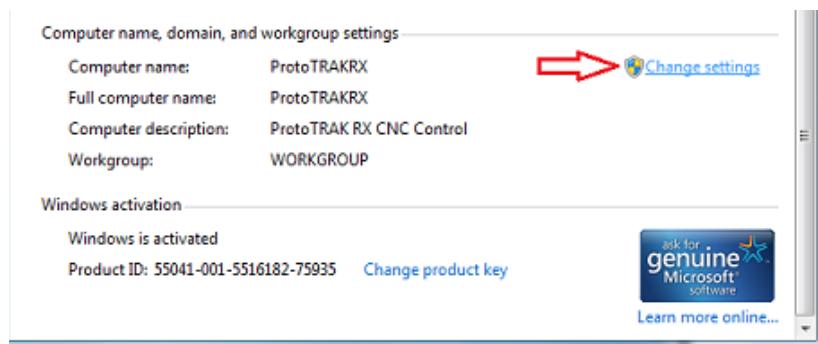
Networking Requirements:
- Setting up the RMX/RLX on a Workgroup for Windows 7
- Straight thru CAT 5 / Ethernet Cable
- Mouse & Keyboard plugged into the RX controller
- Local Admin rights to the PC
- Obtain a DSL Cable router with DHCP Services
- All clicking on the mouse will be done with left button (left click) unless specified to right click
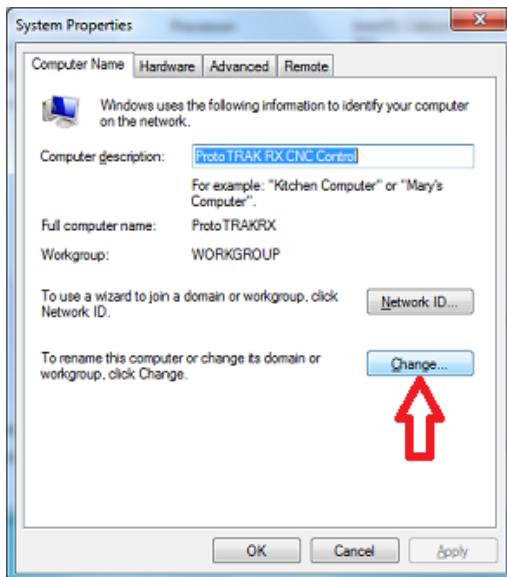
Basic Peer to Peer Network Utilizing Workgroup:
1. In order to establish a Workgroup Network in Windows 7, the computer and control must have the same workgroup name in order to be able to communicate with each other. You'll need to check the Workgroup name on your PC and if you wish to, modify the name as well and then adjust it on the RX side to match in order to be able to network the two together. To do this, perform the following steps:
   a. On your keyboard, press the Windows Logo Key to bring up the Start Menu, locate and right click Computer, then click Properties

   b. Scroll down and locate Computer name, domain, and workgroup settings. Click Change Settings

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
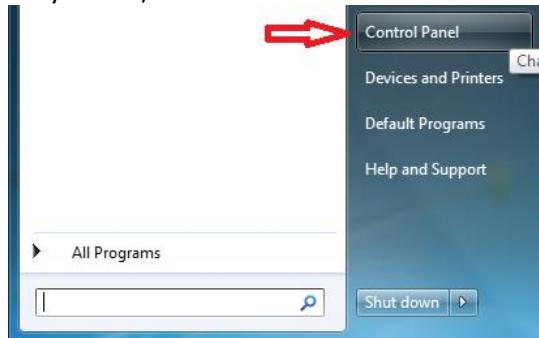
c. On the Computer Name tab, click Change
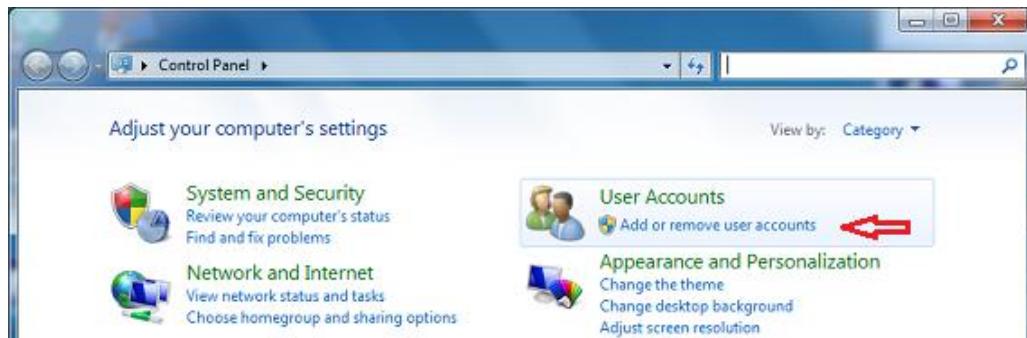
d. Write down the Workgroup name. If you prefer you, you can change the workgroup name here and press OK to change the workgroup name. Write down the name change and follow these same steps to change the workgroup name on the RX control to match the PC. (**Note -** *If you're networking multiple RX Controls, you may need to change the computer name for each one to a unique name. You can change that here as well while changing the Workgroup name.*)

2. **(Optional)** – Secure Network Setup. In order to do a secure network setup requiring a user name and password login, you'll need to create a user account on your PC that can be utilized on the RX control for secure access to a shared folder on your computer. You may also use an existing user account on the PC but it **must** have a password associated with it. To create a user account, do the following:
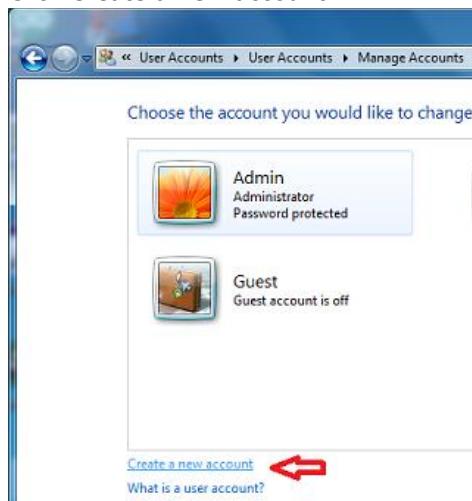
a. On your PC, click Start and then click Control Panel.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
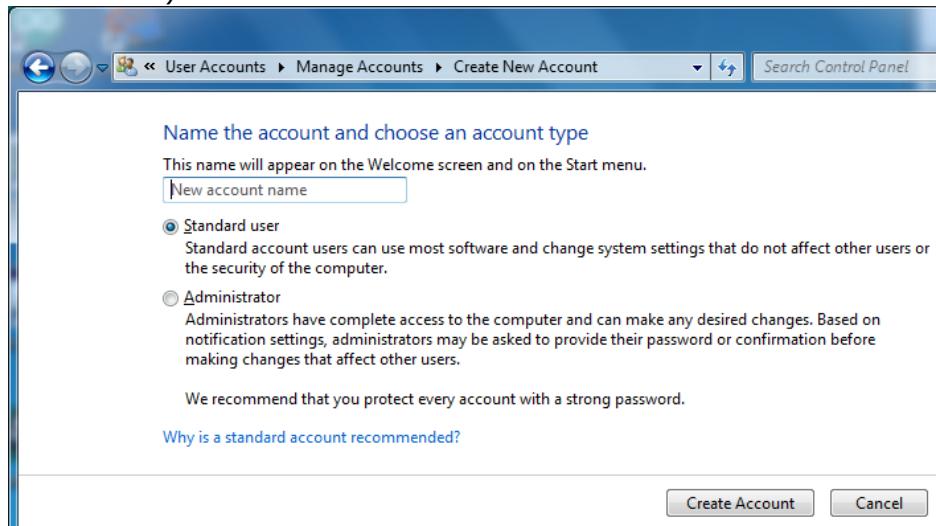
b. Under User Accounts, click Add or remove user accounts.

c. Click Create a new account.

d. Input the user account name and select the user account type (Standard or Administrator) and click Create Account.

e. Once the account is created, click on the account you've created.

f. Click Create a password.
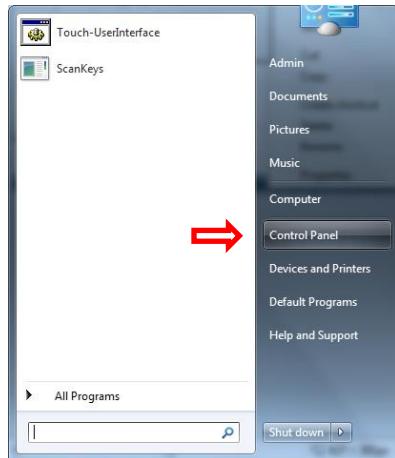
g. Create a strong password that you can easily remember and input a hint as to what your password would be to remind you in case you forget, then click Create Password.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

3. Networking the RX control to the PC – With the workgroup names now changed and securely networking the control, the User account is created, the following are the instruction for creating a mapped network drive and for the control access in order to network the RX control to your PC. Make sure your PC and RX control are connected to the router. If the router has any special requirements, see the instructions for setting up your router or contact the router manufacturer beforehand. If you have any other special network requirements, see your IT Administrator or call into Customer Service 800-367-3165 with any questions for support. There are some common instructions between them so make sure you're following the instruction based on your operating system accordingly.

From the PC: (If you plan to use secure access, have a user account created on the PC for login permission. Make sure the Workgroup name on both PC and RMX are the same.)

   a. Press the Windows Start button on your keyboard and click Control Panel.

   b. Click Network and Internet.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
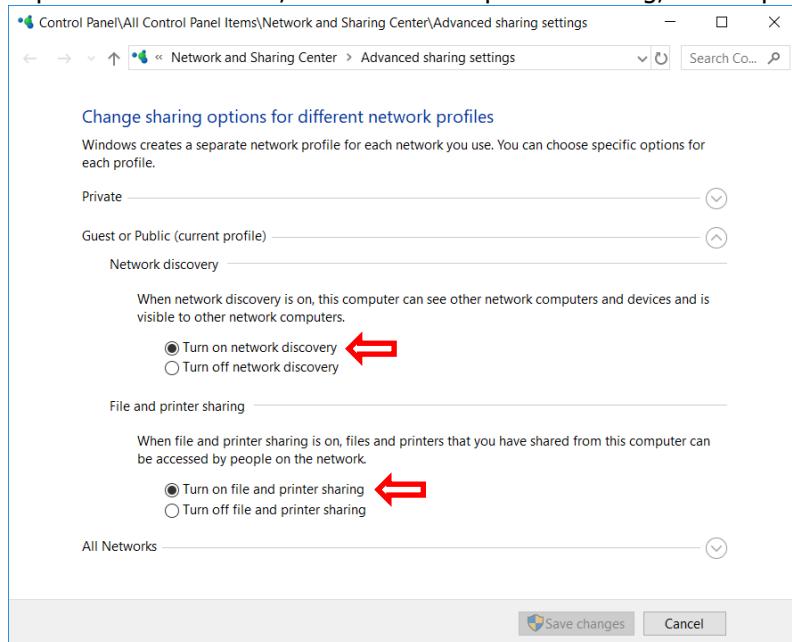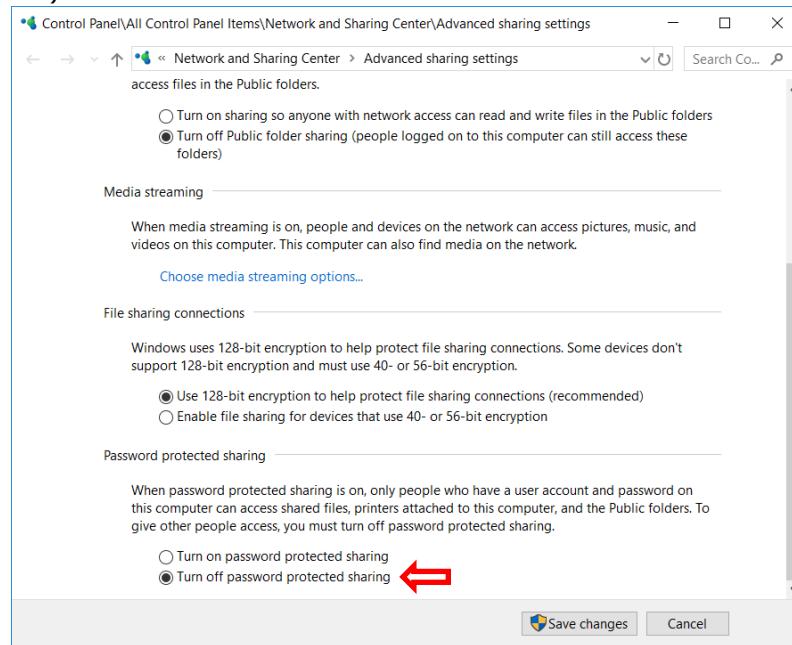
c. Click Network and Sharing Center, then change advanced sharing settings.
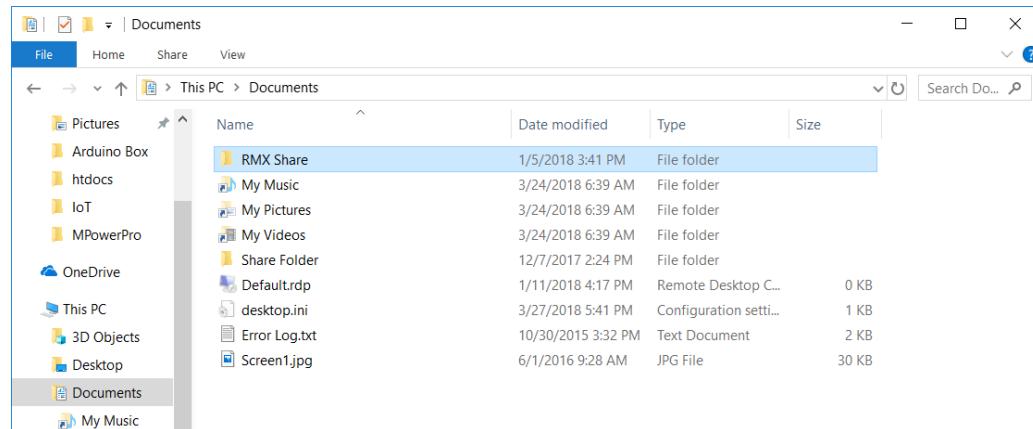
d. Expand Guest or Public, turn on file and printer sharing, then expand All Networks.
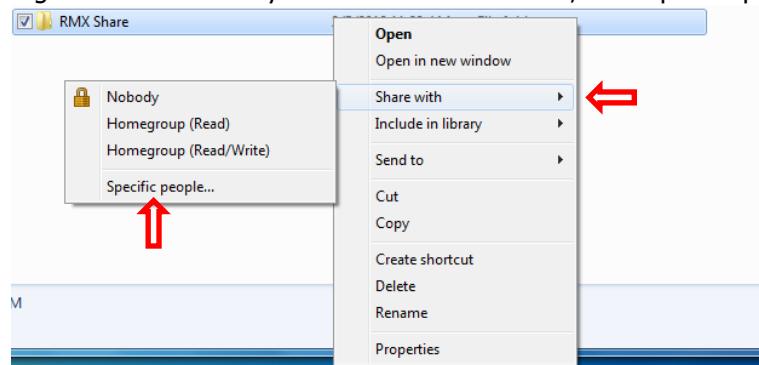
TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

e. Under All Networks, scroll down and turn off password protected sharing (Turn on if you plan to log in with a user account. You must use the login of the user account from the PC.)
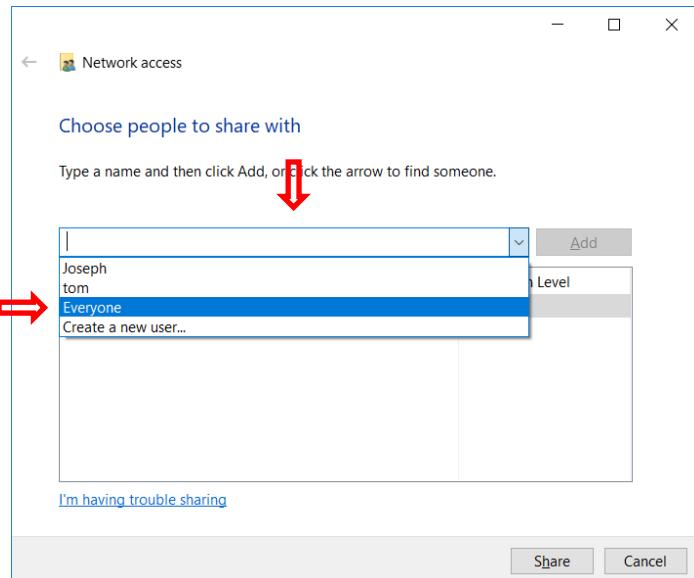
f. Create a shared folder in desired location on the PC.
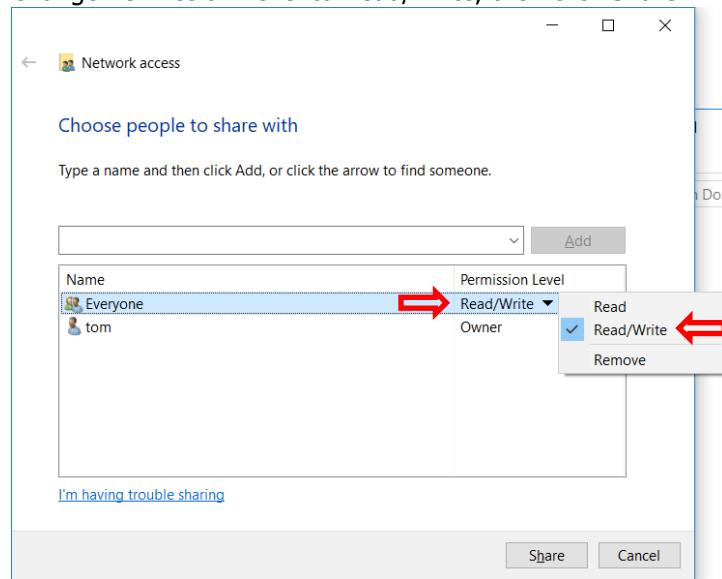
g. Right Click and move your mouse to Share with, click Specific people.

h. Select the drop down and click Everyone.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

i. Change Permission Level to Read/Write, then click Share.

j. On the RMX, press the Windows Start key and click My Computer.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

k.   Click on Map Network Drive.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
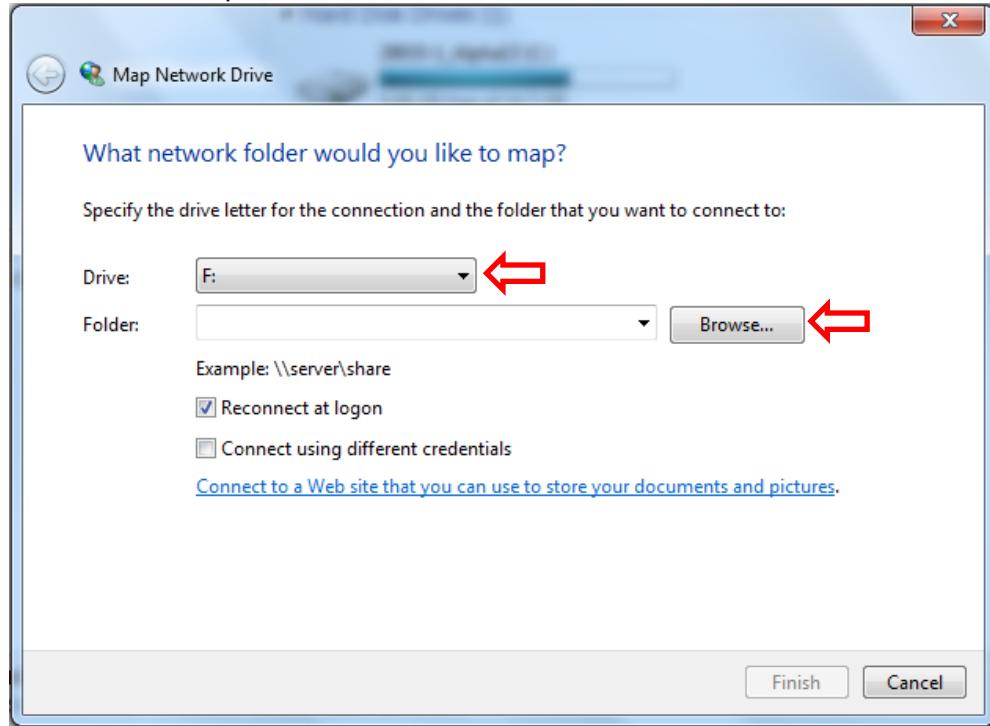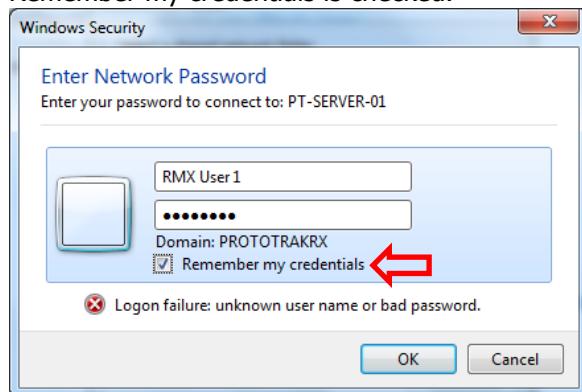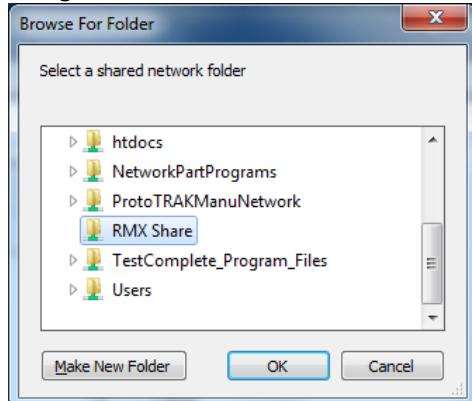
l. Click the Drive drop down menu and select a drive letter. Click Browse.

m. Navigate to the computer your shared folder is on. If you have Password protected sharing on, you'll be required to input your user name and password from User Account you created on your PC (skip this if you have Password protected off). Make sure Remember my credentials is checked.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

n. Navigate to and select the shared folder from the PC you're mapping to and click OK.

o. Click Reconnect at logon then click Finish

14.14.2 Networking – Windows 10
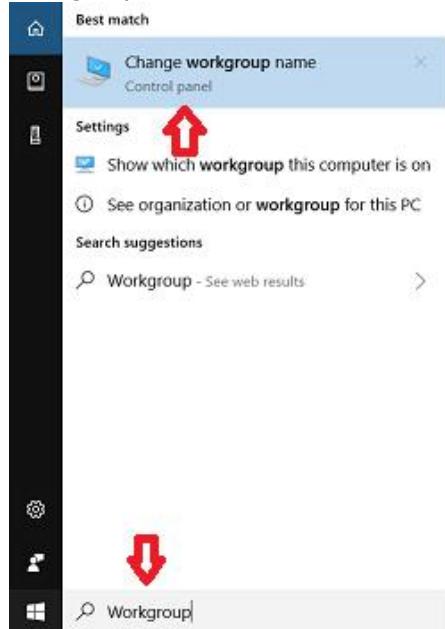
Networking Requirements:
- Setting up the RMX on a Workgroup for Windows 10
- Straight thru CAT 5 / Ethernet Cable
- Mouse & Keyboard plugged into the RX controller
- Local Admin rights to the PC
- Obtain a DSL Cable router with DHCP Services
- All clicking on the mouse will be done with left button (left click) unless specified to right click.
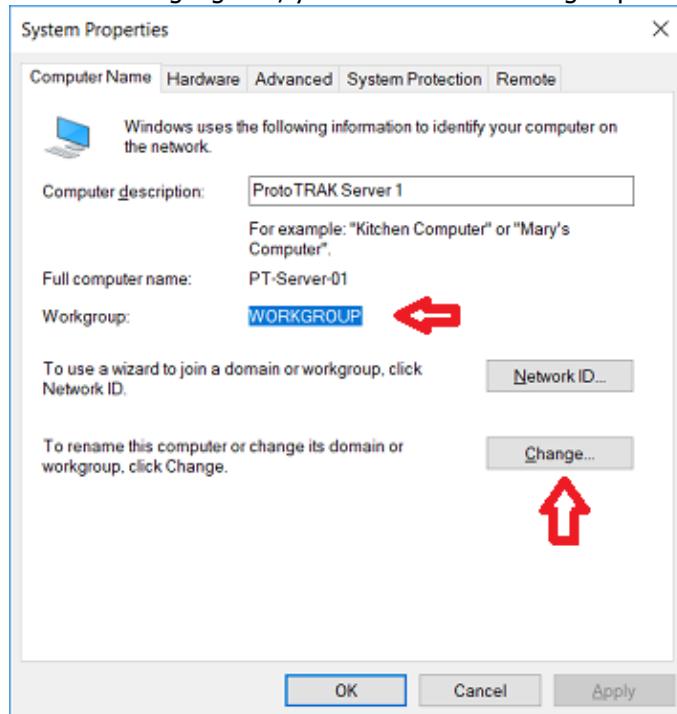
Basic Peer to Peer Network Utilizing Workgroup:
1. In order to establish a Workgroup Network in Windows 10, the computer and control must have the same workgroup name in order to be able to communicate with each other. You'll need to check the Workgroup name on your PC and if you wish to, modify the name as well and then adjust it on the RX side to match in order to be able to network the two together. To do this, perform the following steps:

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
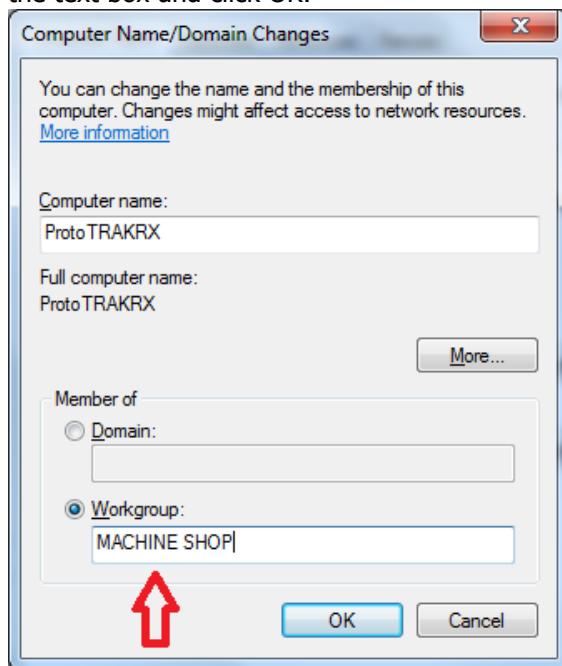
a. On the PC, click on the Cortana Search bar and type in Workgroup, then click Change Workgroup Name.

b. In the area highlighted, you can view the Workgroup names.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
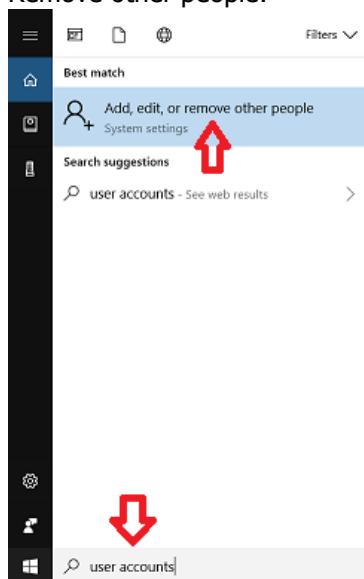
c. If you wish to change it, click Change. Under Workgroup, change the workgroup name in the text box and click OK.
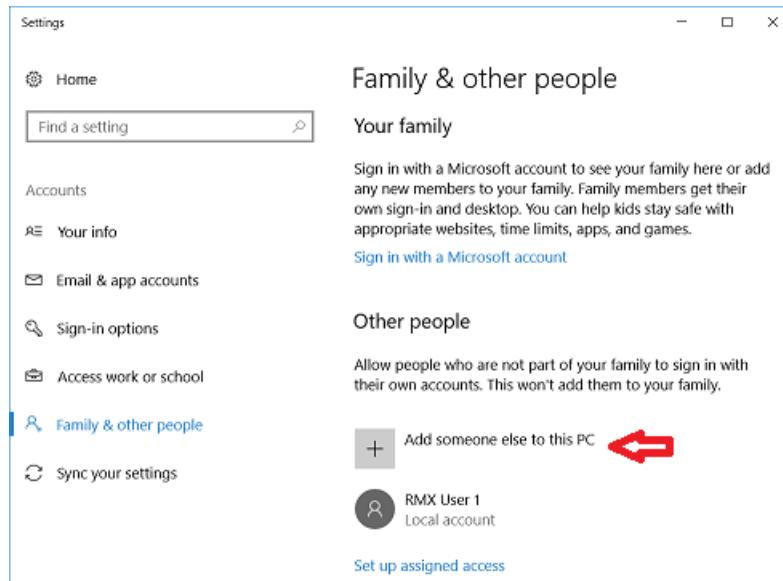
d. Follow steps 1-4 in the previous section 1.2.1 on the RX control to change the Workgroup name to match the Windows 10 workgroup name.

2. **(Optional)** – Secure Network Setup: In order to do a secure network setup requiring a user name and password login, you'll need to create a user account on your PC that can be utilized on the RX control for secure access to a shared folder on your computer. You may also use an existing user account on the PC but it *must* have a password associated with it. To create a user account, do the following:
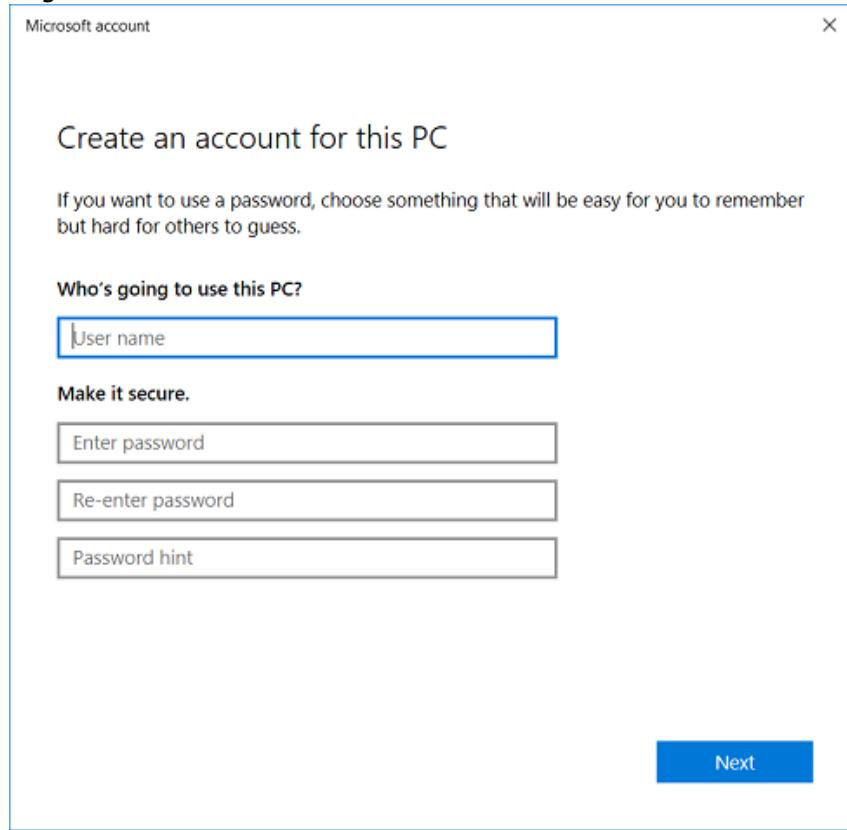
a. Click in the Cortana Search Bar and type in: User Accounts, then click Add, Edit, or Remove other people.

b. Click Add someone else to this PC

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

c. Then type in a desired user name, a strong password that you can remember, and a password hint that will remind you of what that password is in case the password is forgotten and click Next.

3. Networking the RX control to the PC – With the workgroup names now changed and securely networking the control, the User account is created, the following are the instruction for creating a mapped network drive and for the control access in order to network the RX control to your PC. Make sure your PC and RX control are connected to the router. If the router has any special requirements, see the instructions for setting up your router or contact the router manufacturer

beforehand. If you have any other special network requirements, see your IT Administrator or call into Customer Service 800-367-3165 with any questions for support. There are some common instructions between them so make sure you're following the instruction based on your operating system accordingly.

From the PC: (If you plan to use secure access, have a user account created on the PC for login permission. Make sure the Workgroup name on both PC and RMX are the same)
   a. Press the Windows Start button on your keyboard and go to Settings.

   b. Click Network and Internet.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
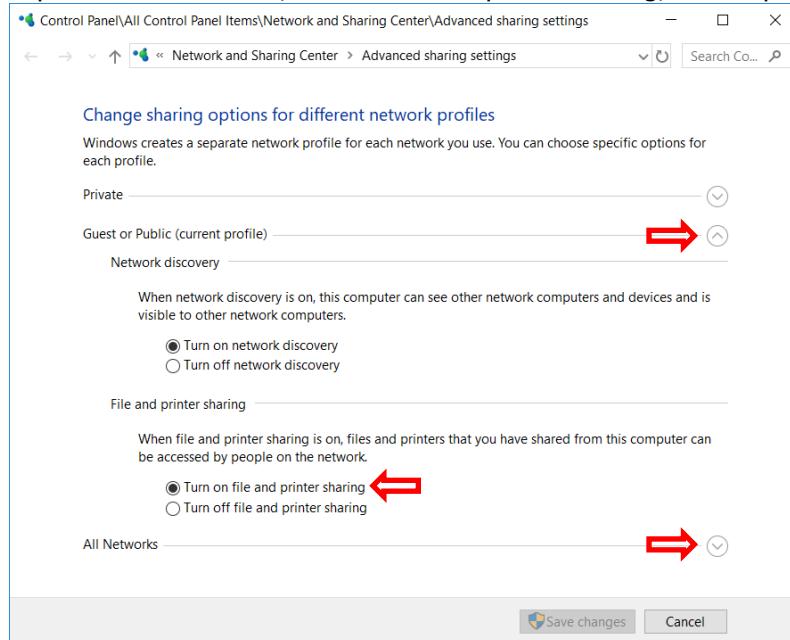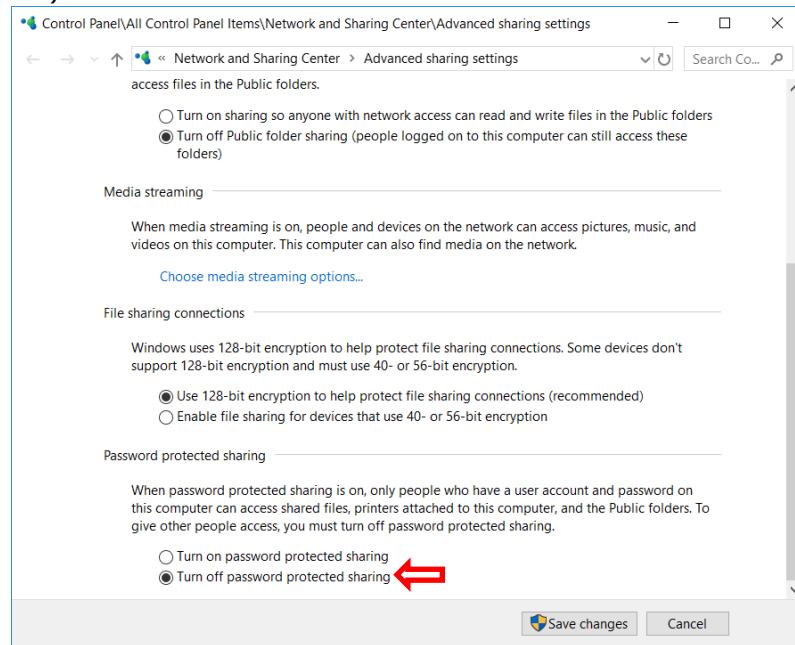
c. Click Sharing options.

d. Expand Guest or Public, turn on file and printer sharing, then expand All Networks.
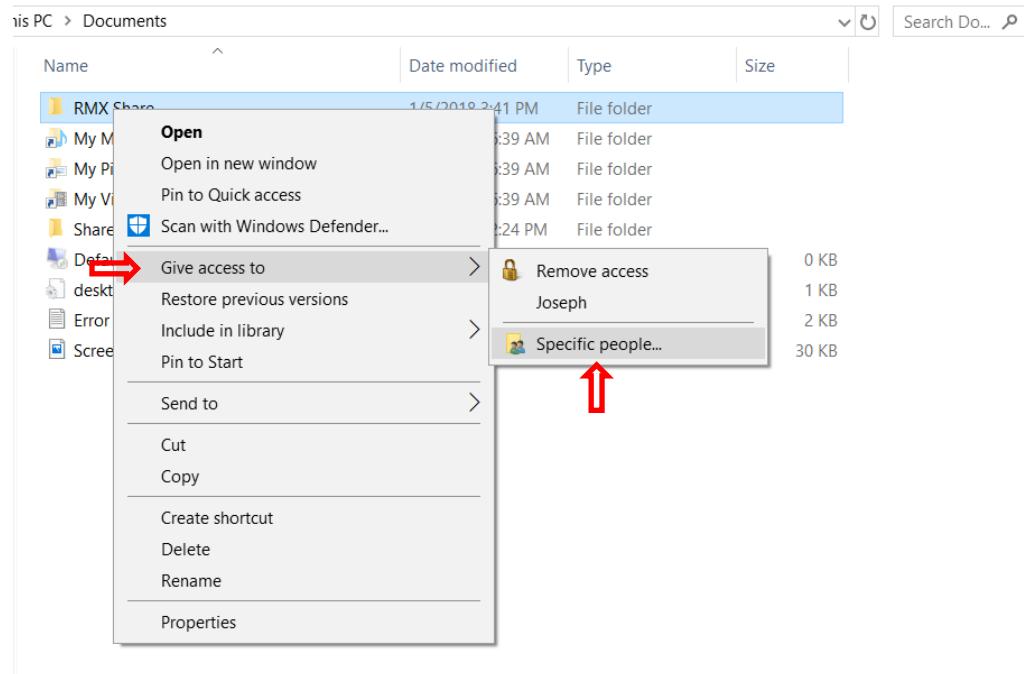
TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

e. Under All Networks, scroll down and turn off password protected sharing (Turn on if you plan to log in with a user account. You must use the login of the user account from the PC.)

f. Create a shared folder in desired location on the PC

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
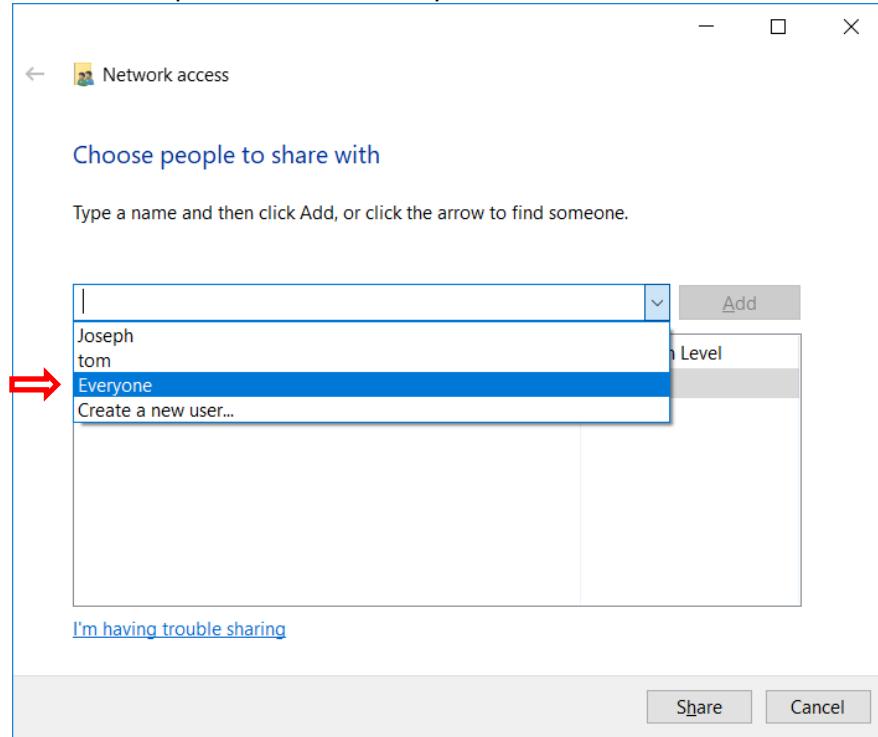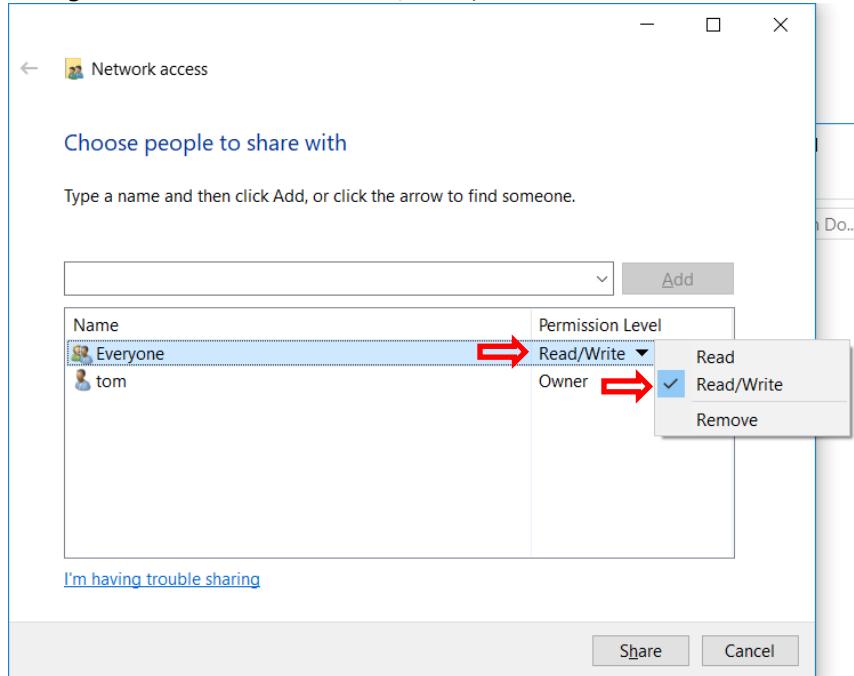
g. Right click and move your mouse to Give access to, then click Specific People
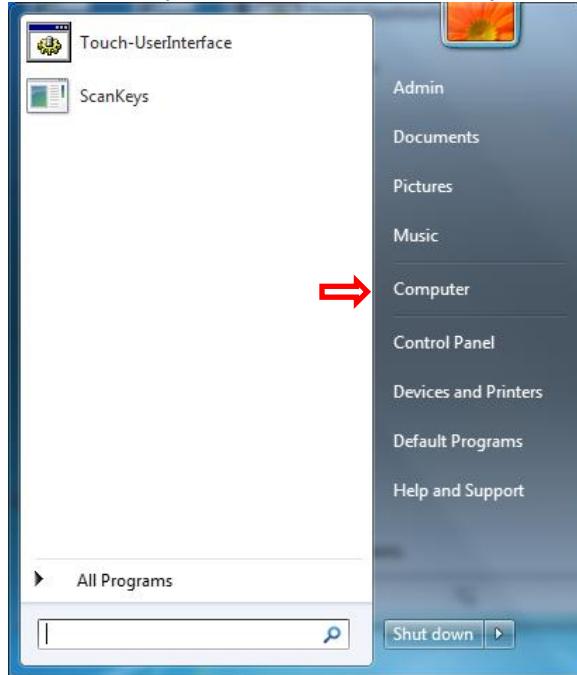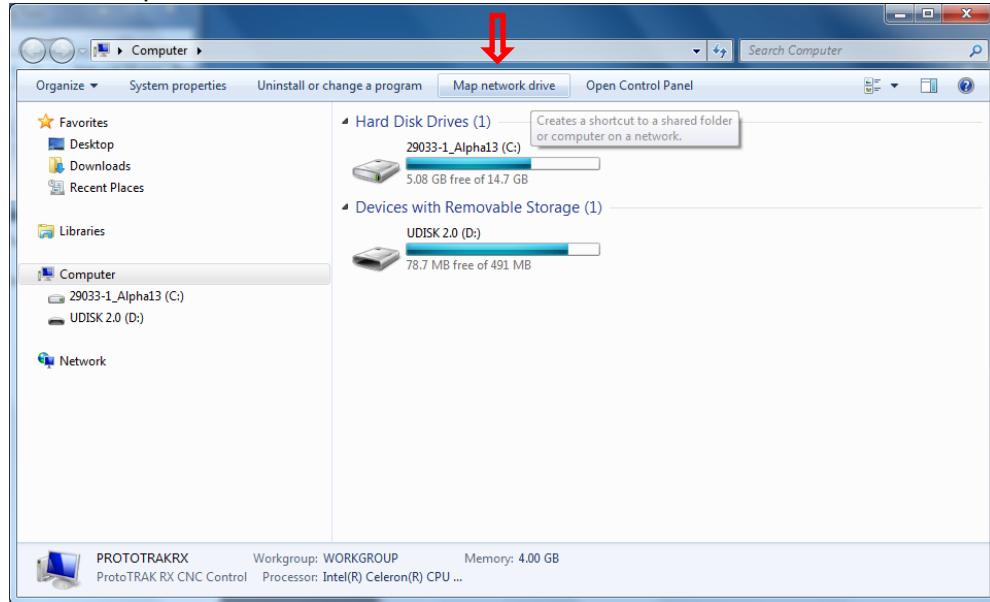
h. Select the drop down and click Everyone.

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

i. Change Permission Level to Read/Write, then click Share.
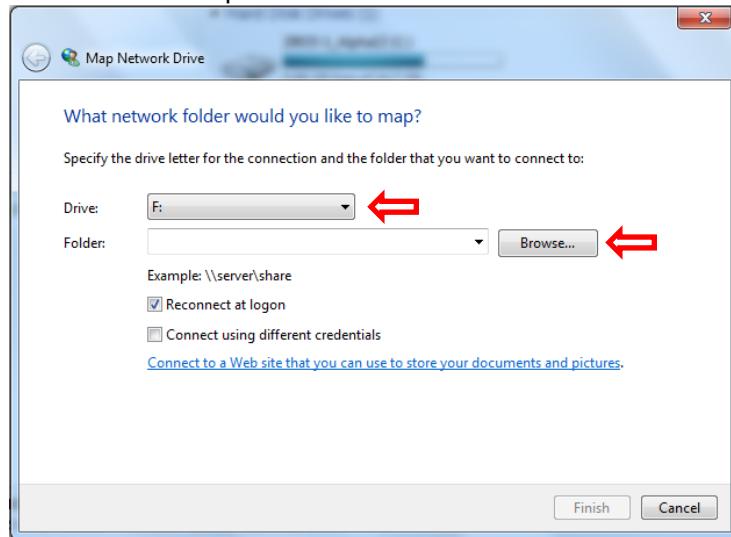
j. On the RMX, press the Windows Start key and click My Computer.

k. Click on Map Network Drive.
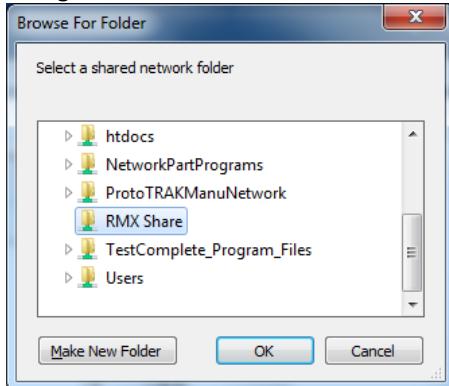
l. Click the Drive drop down menu and select a drive letter. Click Browse.
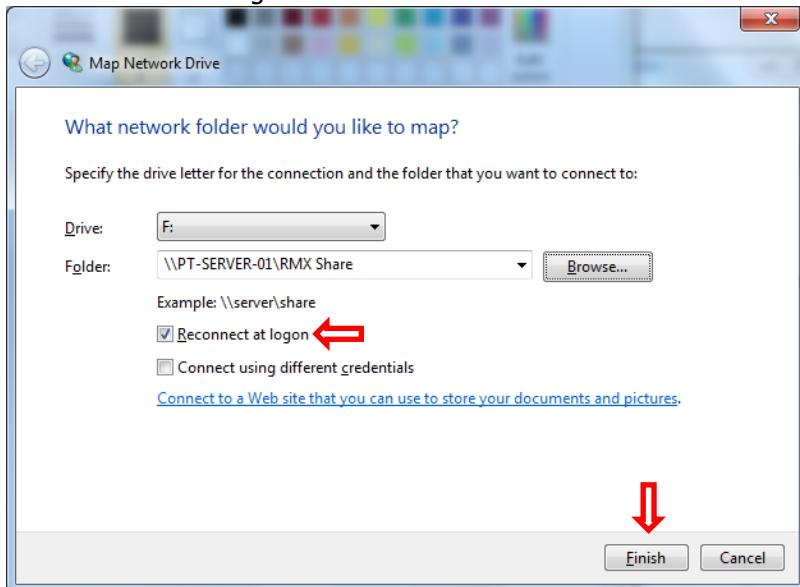
TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

m. Navigate to the computer your shared folder is on. If you have Password protected sharing on, you'll be required to input your user name and password from User Account you created on your PC (skip this if you have Password protected off). Make sure Remember my credentials is checked.

n. Navigate to and select the shared folder from the PC you're mapping to and click OK.

o. Click Reconnect at logon then click Finish

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual

15.0 4th Axis Option – TMC

Upon complete installation of the 4th axis, the option can be activated on the ProtoTRAK screen by following the steps listed below:

1. From the main MODE screen, select:
   a. SET-UP
   b. SERV CODES
   c. A
   d. CODE 318

The following screen will then be displayed:

INFO			MODES
STATUS	A. Advanced Features With Verify B. E Handwheels C. Auxiliary Functions		DRO
TOOL TABLE	**D. 4TH Axis** 1. AutoDesk, Inc. (AutoCad DXF) ACAD .DXF		PROG
EPA	2. Southwestern Industries, Inc. SWI .MX2 3. Southwestern Industries, Inc. SWI .MX3 4. Southwestern Industries, Inc. (IN) F6M .CAM		EDIT
MATH HELP	7. Southwestern Industries, Inc. (GCD) SWI .GCD 8. Southwestern Industries, Inc. (OUT) F6M .CAM 9. Siemens (Parasolid X_T) X_T .X_T		SET-UP
			RUN
DEFAULTS			PROG IN/OUT
KEY BOARD			
CALC			
	INSTALL	RETURN	

2. If the 4th axis option is not yet activated, you will get the chance to do so after selecting the INSTALL softkey by doing the following:
   a. Contact the phone number which appears on the screen.
   b. Provide the TMT Representative with the unique hardware key serial number.
   c. The TMT Representative will then provide an activation password which will enable the 4th axis option.

3. To turn the 4th axis option ON, select:
   a. STATUS info key
   b. 4th AXIS ON

15.1   4th Axis Specifications
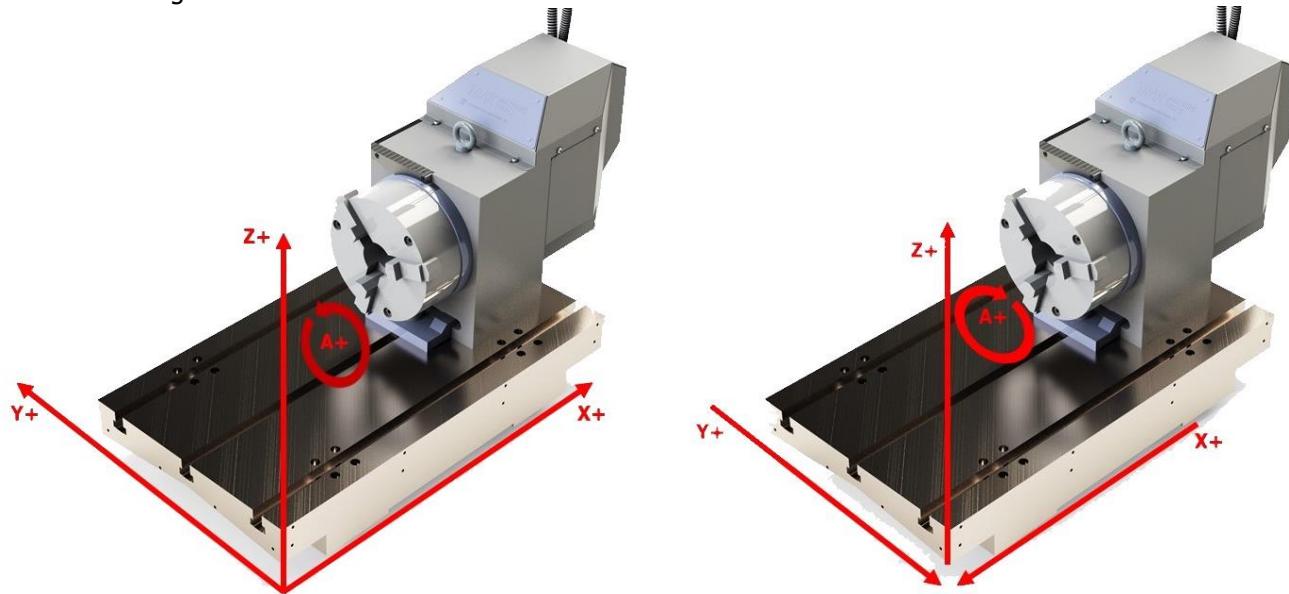
Spindle Diameter	8.07" (205mm)
Chuck Diameter	7.874" (200mm)
Overall Height of 4th Axis	13.82" (351mm)
Centerline Height of Spindle	6.30" (160 mm)
Minimum Resolution of System	0.001º
Maximum RPM	22
Repeatability	+/- 2 arc seconds
Indexing Accuracy	25 arc seconds
4th Axis Weight	190 lbs (86 Kg)
4th Axis Keyway Size	18 mm
Max distance between 4th axis chuck face & tailstock with center*	**TMC5** – 6.26" (160mm) **TMC7** – 14.13" (360mm) **TMC10** – 22.79" (580mm) **TMC12** – 29.88" (760mm) **TMC14** – 41.69" (1060mm)

* Chuck and adapter plate can be removed to add up to 4.65" (113mm) of additional distance. Chuck and adapter plate feed-through hole is 2.28" (58mm).

15.2   Definitions, Terms, and Concepts
15.2.1   4th Axis Direction

It is important to understand the differences in how to command the 4th axis to position in the correct direction. See figures below.

Figure 15.2.1a - Work Coordinates (Positive Direction)

*Figure 15.2.1b - Table Movement (Positive Direction)*

The figure on the left will show you the positive direction for the work coordinate system, i.e., the XYZ and A values that you are programming in relation to your workpiece. The figure on the right shows which direction the machine will actually move in order to get it to go in the positive direction.

The 4th axis is referred to as the A axis, and is measured in degrees. On the DRO screen, it will read from 0.000 degrees to +359.999 degrees. However, when programming, you can refer to negative values, and you can use values of greater than 360 degrees.

Take a look below at figure 15.2.1c. Suppose that point A is our starting point, and A equals +30 degrees ABS. If B was our desired endpoint, then there are actually 4 different ways to program this. By defining B as –30 degrees ABS, or –60 degrees INC, the system will take the first toolpath which is shorter, as shown in figure 15.2.1d. If we define B as +330 ABS, or +300 INC, it will take the long way around, as shown by the second toolpath. Keep in mind that this is meant for 4th axis mill events. If we were only positioning the 4th axis, it will always take the shortest path in order to save time. If you wanted a position event to purposefully take the long way around, such as to get around an obstacle, then you would need to create two position moves, each less than 180 degrees.
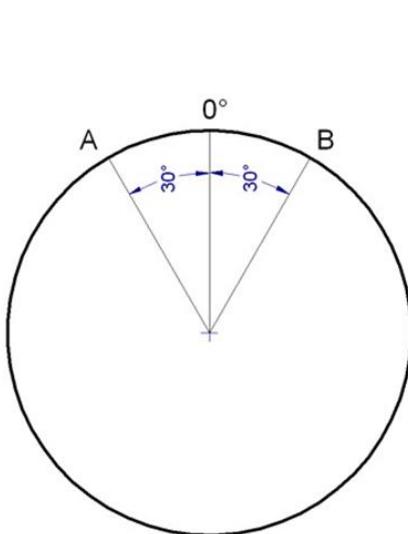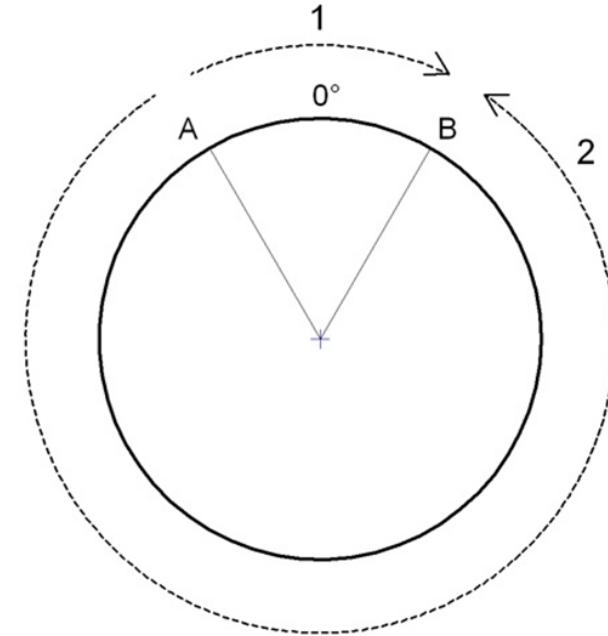
Figure 15.2.1c   **Figure 15.2.1d**

You are also not limited to values that are less than or equal to 360. Take a look at the next figure, 15.2.1e. Imagine that you wanted to cut a groove around a cylinder part, and it needed to wrap around two revolutions. You can program a start point of A 0° ABS, and an end point of A –720° ABS to achieve this (the picture shows the groove spiraling in the negative direction, refer to figure 15.2.1a), while the X axis will simultaneously feed +4.00" across. For more details on how to program this part, see section 15.4.

Figure 15.2.1e

TRAK Machine Tools
Southwestern Industries, Inc.
TRAK Machining Centers and ProtoTRAK RMX CNC Safety, Programming, Operating and Care Manual
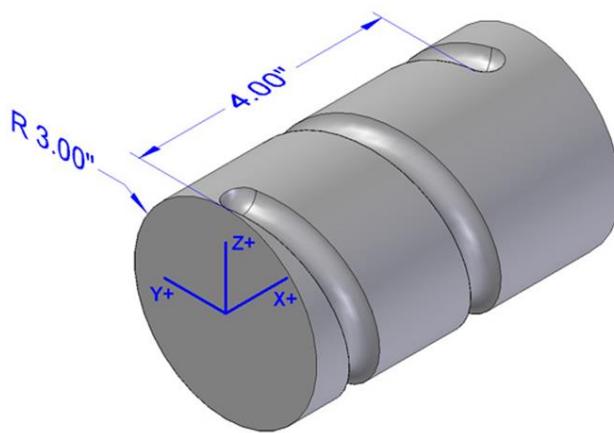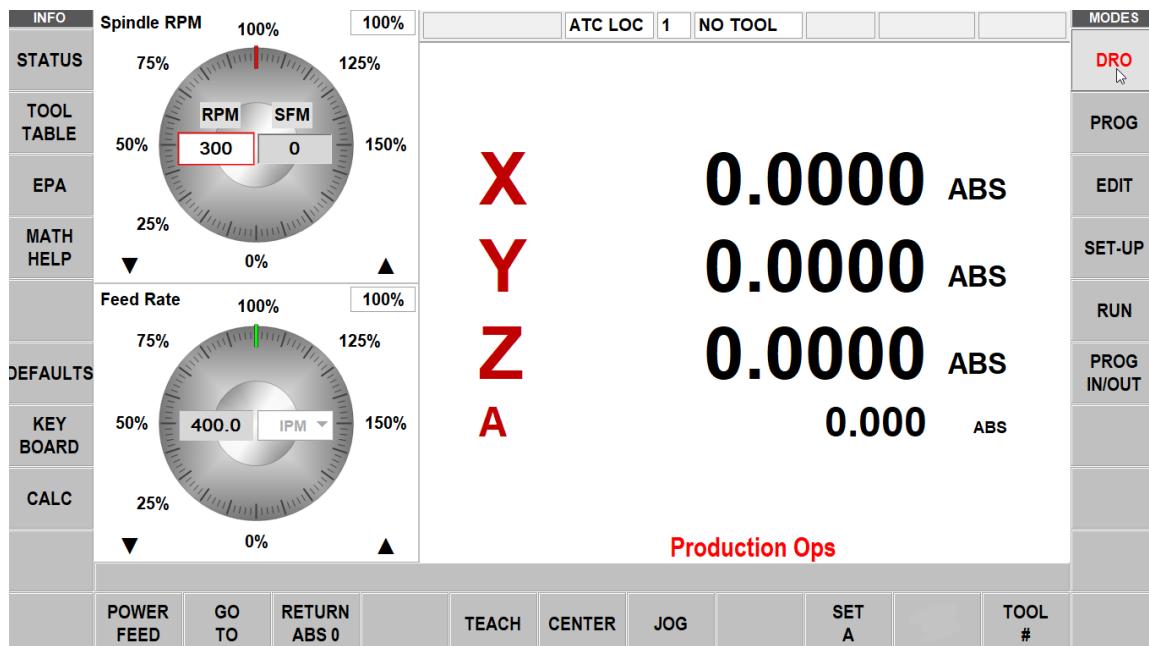
15.2.2 Feedrate

Feedrate for the 4th axis is defined in inches per minute or inches per tooth. When feeding along the A axis, the system will reference from the centerline of the 4th axis. If 20 inches per minute is defined, the system will look at the distance from the current Z axis (tip of the tool, so tool offset needs to be set properly) to the centerline of the 4th axis, and will automatically calculate how fast to turn the A axis in order to achieve the desired feedrate at the tip of the tool.

15.3 DRO Mode

When the 4th axis option is turned on, an extra line will appear for the A axis on the main readout. Additionally, each of the selections in the DRO page will have an option for the A axis.

15.3.1 Jog

To jog the 4th axis, select the EHW hardkey on the Run Panel, then select the 4th hardkey. The electronic handwheel will now jog the 4th axis in either direction.

The resolutions for the JOG speed are as follows:
- .0001" = .001 degree per click
- .0010" = .010 degree per click
- .0100" = .100 degree per click
- FAST = FAST

15.3.2   Go To

Y AND Z OFFSET

Allows the user to view the DRO relative to the centerline of the 4th axis, as defined by service code 506. When manually jogging to the Y or Z zero position, the respective axis will automatically stop at zero. This will allow you to go back to the main DRO screen and set your part's ABS zero relative to this position.

A HOME

Allows the user to re-home the 4th axis, in the event that they need to do so.

POSITION

Define the A axis as a position by selecting the POSITION option under GO TO. When manually jogging to the defined A point, the axis will decelerate and then stop at the defined location.

15.3.3   Return Abs 0

Press RETURN ABS 0, and then GO to return to your part's abs 0 location. The A axis will move back to its 0 by taking the shortest path.
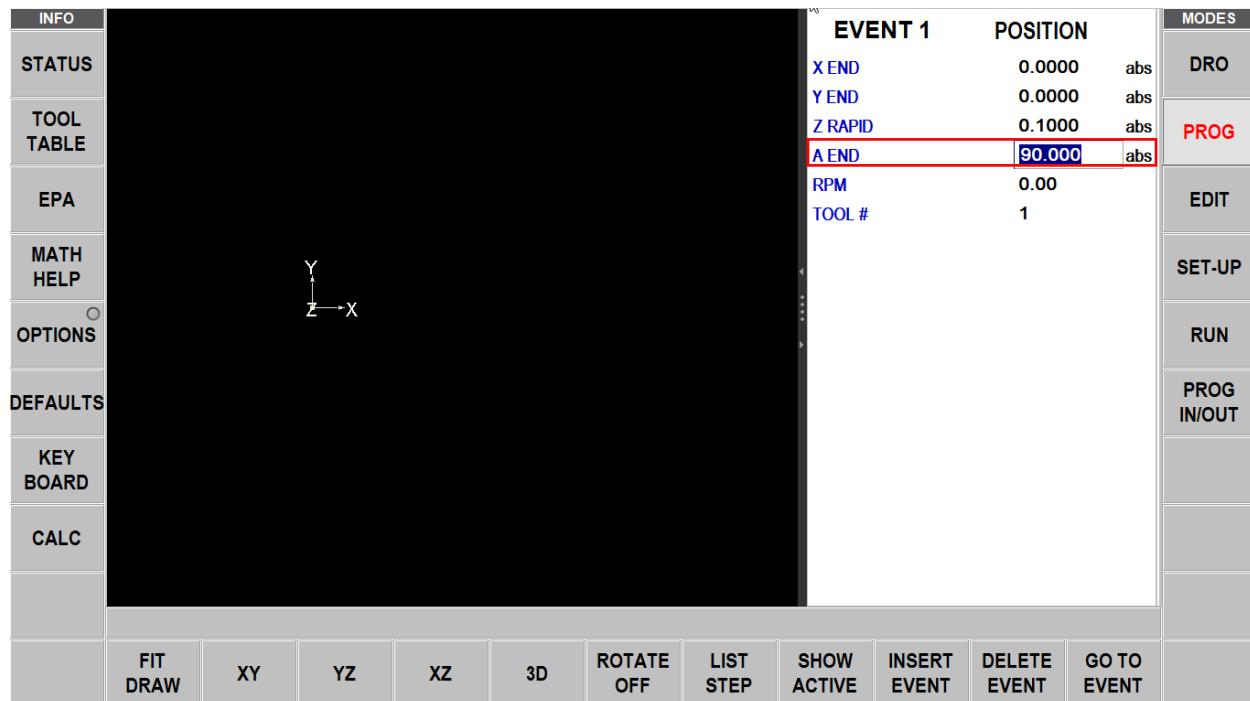
15.3.4   Set A

Press SET A to set your desired part abs position along the A axis.

15.4   Program Mode: Event Types

Once the 4th axis is turned on in MACH SETUP mode, the software will enable new programming options.
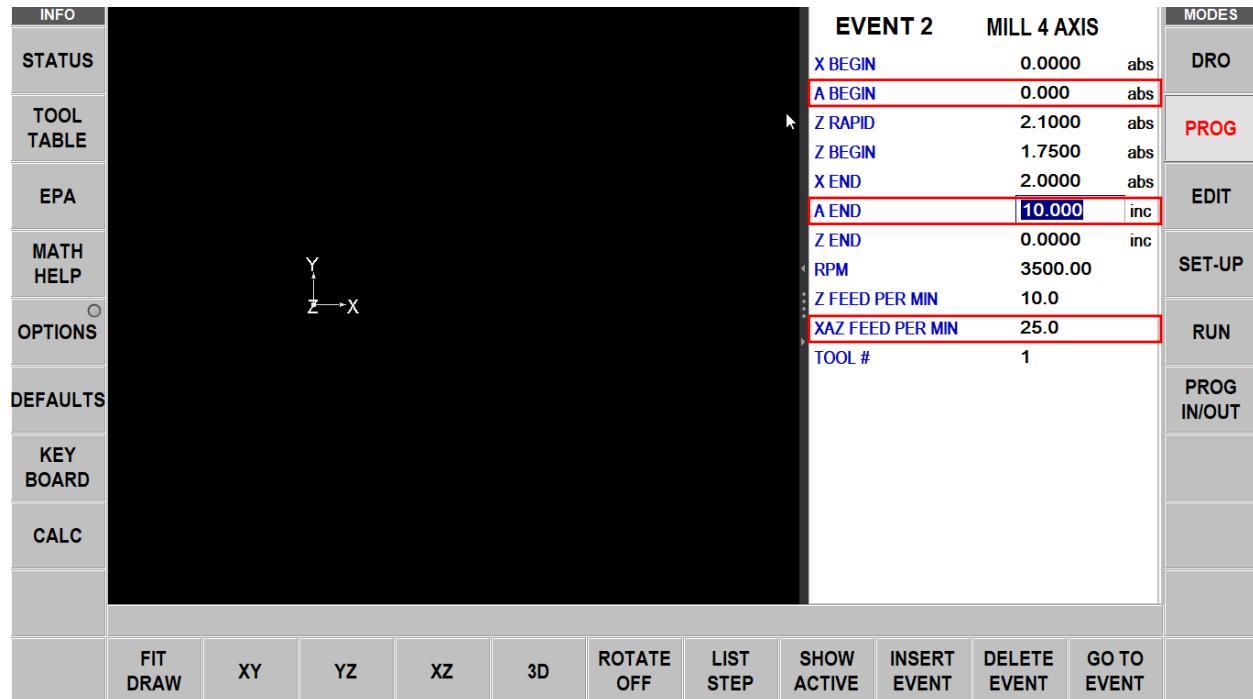
15.4.1   POSN: Position Events

A END - Specifies the value, in degrees, where the 4th axis will rapid to.

15.4.2   Mill 4ᵗʰ Axis

The 4ᵗʰ axis mill event allows you to feed up to all three axis simultaneously, from any X,Z,A beginning point to any X, Z, A ending point. The Y axis is assumed to be at centerline of your part. The Z axis will be the distance from the center of your part, so think of it in terms of radius. Note that there is no cutter compensation available for 4ᵗʰ axis mill events.
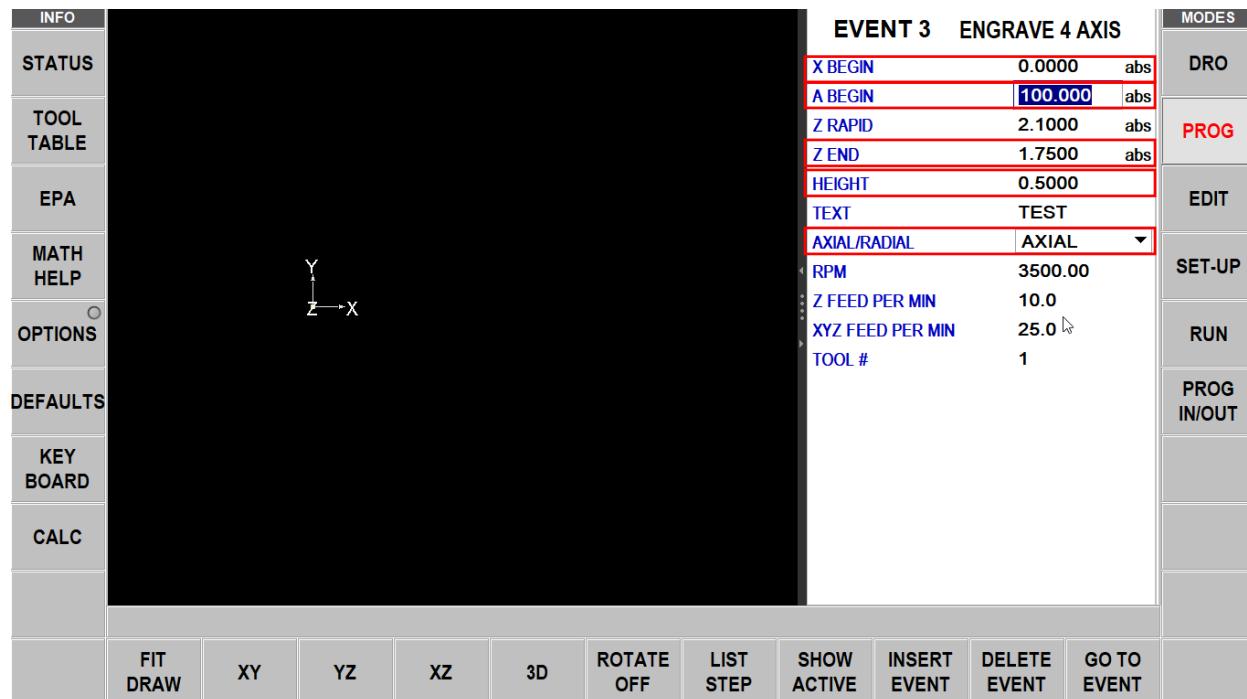
A BEGIN - Specifies the beginning angle of the mill event.

A END - Specifies the ending angle of the mill event.

XAZ FEED PER MIN - Specifies the feedrate of the tip of the tool, in relation to the surface of the workpiece. The feedrate will only be accurate if the part zero ABS is set properly relative to the centerline of the 4th axis, which is defined by service codes 502 and 506.

15.4.3 Engrave 4ᵗʰ Axis

The 4ᵗʰ axis engrave event assumes that you have cylindrical stock, and want to engrave along the surface.

X BEGIN - Defines the starting point on the X axis for the text to be engraved. See figure 15.4.3, and note that even though both radial and axial have the same XA starting point, they span across different areas along the X axis.

A BEGIN - Defines the angle for the starting point of the text to be engraved. See figure 15.4.3 for a visual of both radial and axial text that have the same XA starting point.

Z END - Determines the depth at which the engraving will take place. This distance will be the radius of the part, assuming you have a cylinder stock.

HEIGHT - Defines of the height of the letters, if they were unwrapped onto a flat surface.

AXIAL / RADIAL - Determines which direction the text will wrap around your cylindrical part. See figure 15.4.3.
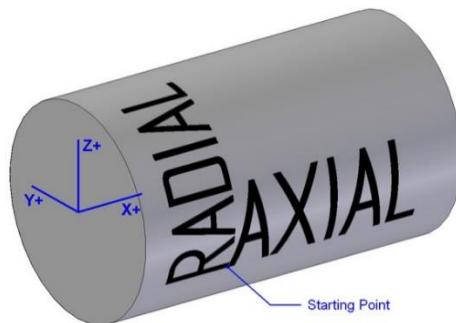

Figure 15.4.3

15.4.4 Rotate ON/OFF

There is a new feature called ROTATE ON or OFF when using the 4th axis in PROGRAM MODE. This feature when selected to ON allows the graphics to rotate the part about the A axis when viewing the program. The feature or surface in question will be highlighted in purple and be rotated so it will be shown as being perpendicular to where the cutting tool would make contact with the part. This feature makes it easier to see the details of your part when you have programmed multiple A axis surfaces.

Pressing the ROTATE ON/OFF button toggles this feature between ON and OFF. For example, when this button reads ROTATE OFF on the screen, it means the graphics will ROTATE each time you press the LIST STEP button. Pressing this button again will turn off the ROTATE feature and the button will read ROTATE ON.

15.5 PROG SETUP Modes
15.5.1 Tool Path

The tool path viewer will now rotate the part about the A axis as you use the STEP feature. The feature or surface in question will be rotated so it will be shown as being perpendicular to where the cutting tool would make contact with the part. This feature makes it easier to see the details of your part when you have programmed multiple A axis surfaces.

15.6 Machine Setup Mode
15.6.1 Service Codes Screen

The following new service codes have been modified or added to the RMX control for the 4th axis unit. For more information on some of the service codes, see the TMC Service Manual.

- **Service Code 318** - Used to turn on the 4th axis option software.

- **Service Code 128** - Add the ability to compensate for A axis backlash. Value entered in decimal degrees. See service manual for more information.

- **Service Code 502 -** Defines the distance from Z home position to the top of the table, using your base tool as reference.

- **Service Code 505** - Added a prompt for the A axis motor index

- **Service Code 506** - Set sets the Y and Z offsets for the 4th axis unit. The following describes what each offset is.
    - **Y Offset** - The distance along the Y axis from Y home to the centerline of the 4th axis.

    - **Z Offset** - The distance along the Z axis from the top of the table to the centerline of the 4th axis.
      Note - See the TMC service manual for how to set the Z offset location.

15.6.2 4th Axis ON/OFF

This toggle switch enables and disables the use of the 4th axis unit, as well as the programming options available to the user. Whenever the 4th axis option is turned ON or OFF, the machine requires a power cycle. After turning on the 4th axis, the user will need to home the machine again so that the 4th axis gets homed as well. The 4th axis option cannot be on while in 2 axis mode, you will be prompted to turn the 4th axis OFF.

15.7 PROG I/O

If opening a program that contains 4th axis specific events, and the 4th axis option is not activated on the machine, the system will display an error message. The user must enable the option using service code 318. If the 4th axis option is turned off, and the user tries to open a program containing 4th axis events, the machine will display a message asking the user to turn ON 4th Axis. Similarly, when a user tries to open a 2 axis program when 4th axis option is turned ON, the machine will display a message asking the user to turn OFF 4th Axis.

15.7.1 4th Axis G-Code Notes

The ProtoTRAK RMX is capable of running 4th axis G-Code. In addition to all the codes found under section 14.12 and 14.13, please use the additional notes below when trying to define your post processor.

- The 4th axis is defined by A, and is always assumed to be parallel to the X axis.

- Only G00 and G01 commands support movement along the A axis. The format will look like the following:
  - G00 X1.0 Y2.0 Z3.0 A4.0
  - G01 X5.0 Y6.0 Z7.0 A8.0

- The feedrate along A must be defined in inches per minute. Feedrate is calculated in relation to the centerline of the part, which our system is always keeping track of using service code 506.

- For best results when running GCD, the Z axis g-code should be output in relation to center of part.

- The angle for A axis can be defined in incremental or absolute values, using G90 / G91, can be positive or negative, and can exceed 360.

- G41 and G42 are not supported when feeding along the 4th axis. G40 must be used.

15.8 4th Axis Setup Notes

- See section 6 of the TMC service manual for important tips for taking the 4th axis unit on and off of the machine when not in use. The 4th axis unit weighs about 190 lbs, so please use a lift to remove the unit from the machine. The 4th axis unit comes with an eyebolt that can be used for lifting purposes. You will also need to disconnect two cables before removing the unit.

  Note – *When the 4th axis is removed from the machine, please make sure to install the provided jumper in the electrical cabinet on the computer module. See section 6.3 of the service manual for a picture*

- The 4th axis comes with a standard 3 jaw chuck and adaptor plate. If you need to remove this chuck, please see the service manual for instructions. It should be noted that the adaptor plate and chuck will need to be trammed in when it is put back on.

TRAK Machine Tools
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/ProtoTRAK	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 product, subsystem or component proves to be defective in workmanship and fails within the warranty period, 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.

Warranty Disclaimers

- 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/electronic systems and must be given the reasonable care that these types of products require. Evidence that the product does not receive adequate Preventative Maintenance may invalidate the warranty. Excessive chips built up around ballscrews and way surfaces is an example of this evidence.

- Accidental damage, beyond the control of SWI, is not covered by the warranty. Thus, the warranty does not apply if a product has been abused, dropped, hit or disassembled.

- 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.

- Warranty does not cover wear items that are consumed under normal use of the product. These items include, but are not limited to: windows, bellows, wipers, filters, drawbars and belts.

F10302 | Rev: 020620