

TORMACH® PCNC 440™

OPERATOR'S MANUAL



IMPORTANT! Read and understand all operator's manual safety precautions and instructions before attempting PCNC 440 installation, operation, or maintenance.

Questions or comments?

Please email us: info@tormach.com
All rights reserved.

 **TORMACH®**

UM10372_PCNC440_Manual_0219A
Document Part Number: 35672
©2018 Tormach Inc.

PREFACE

SAVE THESE INSTRUCTIONS!

This manual contains important safety warnings and operating instructions for the PCNC 440 mill. Refer to these instructions before attempting installation, operation or maintenance. Keep these instructions together with your PCNC 440 mill so they are readily accessible. The most recent version of this manual is available at: www.tormach.com/documents

Read Before Operating

Read and follow all warnings, cautions, and operating instructions before operating this machine. Failure to do so could result in voided warranty, property damage, serious injury or death.

Symbol	Description	Example
	WARNING! Indicates a hazard which, if not avoided, could result in death or serious injury.	WARNING! Ejection Hazard: Tools and workpieces must be clamped properly. Failure to do so could result in death and/or serious injury.
	CAUTION! Indicates a hazard which, if not avoided, could result in injury or mill damage.	CAUTION! Sharp Objects: Be sure to wear gloves when uncrating mill. Failure to do so could result in serious injury.
IMPORTANT!	IMPORTANT! Addresses important practices not related to personal injury.	IMPORTANT! Damage to mill may occur if motor weight is supported by motor wires.
NOTE:	NOTE: Indicates additional information, clarification, or helpful hints.	NOTE: Allocate additional space to allow access to rear of mill for maintenance or repairs.

Safety Overview

Any machine tool is potentially dangerous. The automation inherent in a CNC machine presents added risk not present in a manual machine. Tormach CNC machines can deliver sufficient force to break brittle tools, crush bones, and tear flesh.

This manual provides guidance on safety precautions and techniques, but because the specifics of any one workshop (or other local conditions) can vary greatly, Tormach accepts no responsibility for machine performance or any damage or injury caused by its use. It is your responsibility to ensure you understand the implications of what you are doing and comply with any legislation and codes of practice applicable to your city, state or nation.

Machine Safety

Safe operation of the machine depends on its proper use and the precautions taken by the operator. Read and understand this manual. Only trained personnel – with a clear and thorough understanding of its operation and safety requirements – should operate this machine.

General Safety:

- Wear OSHA-approved safety glasses, safety shoes, and ear protection.
- Remove loose-fitting clothing, neckties, gloves, and jewelry.
- Tie up long hair or secure under a hat.
- Never operate a machine after consuming alcohol or taking medication.
- Keep work area well lit and deploy additional lighting, if needed.

Operational Safety:

- Understand CNC machines are automatically controlled and may start at any time.
- Do not leave machine unattended during operation.
- Always power off machine when not in use.
- Never operate with unbalanced tooling or spindle fixtures.
- Remove all tools (wrenches, chuck keys, etc.) from spindle and machine surface before starting operations; loose items can become dangerous projectiles.
- Use adequate work clamping; loose workpieces can become dangerous projectiles.
- Protect your hands. Stop machine spindle and ensure machine motion has stopped before:
 - Reaching into any part of the machine motion envelope
 - Changing tools, parts or adjusting the workpiece
 - Changing belt/pulley position
 - Clearing away chips, oil or coolant; always use a chip scraper or brush
 - Making an adjustment to part, fixture, coolant nozzle or when taking measurements
 - Removing protective shields or safeguards; never reach around a guard
- Keep work area clear of clutter as machine motion can occur when keys are accidentally pressed or objects fall on keyboard, resulting in unexpected motion.
- Position clamping attachments clear of tool path. Be aware of workpiece cutoffs that could be cut free during operations and become dangerous projectiles.
- Always use proper feeds/speeds, as well as depth/width of cut to prevent tool breakage.

PREFACE

- Check for damaged tools/workpieces and cease operations if detected; replace before re-starting operations as these can become dangerous projectiles. Never use longer or larger tools than necessary.
- Chips and dust from certain materials (e.g., magnesium) can be flammable. Fine dust from normally non-flammable materials may be flammable or even explosive.
- Chips, dust, and vapors from certain materials can be toxic. Always check the Materials Safety Data Sheet (MSDS) for each material.

IMPORTANT! *It is the responsibility of the employer/operator to provide and ensure point of operation safeguarding per the following:*

- OSHA 1910.212 – General Requirements for All Machines
- OSHA 1910.212 – Milling Machines, Point of Operation Safeguarding
- ANSI B11.22-2002 Safety Requirements for Turning Centers and Automatic Numerically Controlled Turning Machines
- ANSI B11.TR3-2000 Risk Assessment and Risk Reduction – A Guideline to Estimate, Evaluate, and Reduce Risks Associated with Machine Tools
- Safety Requirements for Construction, Care, and Use of Drilling, Milling and Boring Machines (ANSI B11.8-1983). Available from American National Standards Institute, 1430 Broadway, New York, New York 10018
- Concepts and Techniques of Machine Safeguarding (OSHA Publication Number 3067). Available from The Publication Office – OSHA, U.S. Department of Labor, 200 Constitution Avenue, NW, Washington, DC 20210

Electrical Safety



WARNING! Electrical Shock Hazard: Be sure to power off machine before making any electrical modifications. Failure to do so could result in death and/or serious injury.

Input Power: The PCNC 440 has one 115 V power input which is capable of delivering lethal electrical shocks. Care should be exercised when working inside the electrical cabinet.

Grounding: The power input must be grounded. Do not assume during installation that a wall outlet is properly grounded. Check continuity between the machine frame and true earth ground (metal water pipe or similar) to ensure a good ground connection.

Ground Fault Interrupter: A Ground Fault Interrupter or GFI (also known as a Residual Current Circuit Breaker or RCCB) outlet must not be used to supply power to the 115 VAC power input as it will interfere with the spindle drive electronics.

Electrical Cabinet: Never operate the mill with the electrical cabinet open. Never allow a coolant pump to operate with the electrical cabinet open. Do not allow the coolant system to flow coolant directly at the electrical cabinet or the operator box. Neither the electrical cabinet nor the operator box controls are hermetically sealed against liquids.

Electrical Service: Certain service and troubleshooting operations require access to the electrical cabinet while power is on. Only qualified electrical technicians should perform such operations.

Retained Electrical Power: Electronic devices within the electrical cabinet may retain dangerous electrical voltage after the power is off.

Support

Tormach provides no-cost technical support to our customers through multiple channels. The quickest way to get the answers you need is normally in this order:

- Refer to this operator manual first
- Reference related documents at: <http://www.tormach.com/documents.html>
- Email: info@tormach.com
- Phone: 608-849-8381 x2001, Monday through Friday 8 a.m. to 5 p.m. (central standard time)
- Fax: 209-885-4534

Scope and Intellectual Property

This document is intended to provide sufficient information to allow you to install, setup, and use your Tormach PCNC 440 mill. It assumes that you have appropriate experience and/or access to training for any computer-aided design/manufacture software to use with the machine.

Tormach Inc. is dedicated to continual improvement of its products, so suggestions for enhancements, corrections, and clarifications are welcome.

The right to make copies of this manual is granted solely for the purpose of training courses related to, evaluation of and/or use of the PCNC 440. It is not permitted, under this right, for third parties to charge for copies beyond the cost of printing.

Every effort has been made to make this manual as complete and as accurate as possible but no warranty or fitness is claimed or implied. All information provided is on an *as is* basis. The authors, publisher, and Tormach Inc. shall not have any liability for, or responsibility to, any person or entity for any reason for any loss or damage arising from the information contained in this manual.

Tormach, PCNC 440 Personal CNC, Tormach Tooling System (TTS), and PathPilot are trademarks or registered trademarks of Tormach Inc. If other trademarks are used in this manual, but not acknowledged, please notify Tormach Inc. so this can be remedied in subsequent editions.

Tormach milling machines and accessories are covered by one or more of the following U.S. Patents: 7,386,362, D606,568, D612,406, D621,859 and Patent(s) Pending.

PREFACE

Intended Use Statement

The PCNC 440 is intended for use as a general purpose CNC milling machine. The intended use includes cutting conventional (non-abrasive) materials such as unhardened mild or alloy steels, aluminum, plastics, wood, and similar materials (or any other material that can be cut with a rotating cutter).

Outside Scope of Intended Use

Applications for the equipment or modifications of the equipment outside of the *Intended Use Statement* are supported through consulting engineering and excluded from Tormach's no-cost technical support.

All of the technical information and insight required to support variations from the intended use cannot possibly be foreseen. If the extensive documentation provided does not supply all the information you need, we can provide additional information and engineering support required for your project on a consulting engineering basis. If you have your questions well organized, we can normally provide all the information you need in short order. Consulting engineering is done by electrical and mechanical engineers and billed at current hourly rates.

All warranties for Tormach equipment are voided through modification to the equipment or use outside of the intended use. Individuals or companies involved with modifying the equipment or applying the products assume all consequent liability.

Performance Expectations and Cutting Ability

The following table summarizes the cutting performance envelope of the PCNC 440 mill:

Spindle Speed Range	300-10,000 RPM
Spindle Power Rating	3/4 hp (600 W)
Feed Rate Range	0-135 IPM (X, Y) 0-110 IPM (Z)

The PCNC 440 is capable of cutting any material that can be cut with a rotating cutter at or near its recommended feeds and speeds. As with any machine tool, care should be exercised so that programmed cuts do not exceed the maximum available spindle horsepower. Small diameter cutters may perform better with use of a companion spindle or RPM multiplier.

Resolution, Accuracy, and Repeatability

The following table summarizes resolution, accuracy, and repeatability of the PCNC 440 as delivered.

Resolution of Motion (minimum discrete positional move)	0.0001"
Positional Accuracy ¹	≤ 0.0016" per 10"

¹ Includes additional contributing factors such compressibility of bearings, ball screw windup, friction, etc.

Each PCNC 440 ships with a Certificate of Inspection. This report details each of the quality assurance measurements performed at the factory by a Tormach quality assurance team member prior to approving shipment of each machine.

A sample certificate of inspection and more information on quality assurance measurements is available at: http://www.tormach.com/quality_overview.html

In practice, accuracy and repeatability are heavily influenced by the techniques used by the machinist. A skilled machinist can often deliver accuracy that exceeds the accuracy specified by the machine builder, while an inexperienced machinist may have difficulty delivering the expected accuracy. With this understanding, we cannot tell you what accuracy you will be able to achieve in your own work. Nevertheless, the accuracy specified by a machine builder remains an important reference point.

Nomenclature

The following table summarizes the typographical nomenclature this manual uses:

<i>Software Control</i>	Refers to a Software Control i.e., an on-screen button.
<i>Hardware Control</i>	Refers to a physical button or switch on the mill.
G-code (e.g., G01X34.8)	Used to show G-code.
<i>Key name</i> (i.e., <i>Enter</i>)	Tells you to press the indicated key.
<i>Button name</i> (i.e., <i>Stop</i>)	Tells you to press the indicated button.

PREFACE

Table of Contents

1. Overview.....	19
1.1 Specifications.....	20
2. Site Planning and Prep.....	21
2.1 General Site Requirements.....	21
2.1.1 Space Requirements.....	21
2.2 Electrical Requirements.....	21
2.2.1 Grounding.....	21
2.2.2 Plug Pattern.....	21
2.2.3 Ground Fault Interrupter (GFI) Use.....	21
2.2.4 Electrical Noise.....	22
2.2.5 Options for Electrically Non-conforming Sites.....	22
3. Installation.....	23
3.1 Receiving, Uncrating, and Initial Inspection	23
3.1.1 Shipment Arrival.....	23
3.1.2 Moving the Crate.....	23
3.1.3 Initial Uncrating.....	24
3.1.4 Shipping Damage or Shortages.....	24
3.2 Installation Sequence.....	24
3.3 Basic Installation Procedure.....	24
3.3.1 Stand Assembly.....	24
3.3.2 Remove Pallet Items.....	25
3.3.3 Lift and Move Mill.....	25
3.3.3.1 Remove Mill from Pallet.....	25
3.3.3.2 Lift Mill.....	25
3.3.4 Attach Mill to Stand.....	27
3.3.4.1 Non-Tormach Stands.....	27

3.3.5 X-axis Motor Cover Plate.....	27
3.3.6 X-axis Limit Switch Cover Plate.....	27
3.3.7 Manual Oiler.....	27
3.3.8 Flood Coolant System.....	27
3.3.9 Install PathPilot Controller.....	28
3.4 Essential Controls Overview.....	31
3.4.1 E-stop, Start, Reset, and Main Disconnect.....	31
3.5 Power Off/Power On Procedure.....	33
3.6 Initial PathPilot Controller Configuration.....	35
3.7 Validate Basic Installation.....	
3.7.1 Verify Spindle Function.....	
3.7.2 Verify Limit Switch Function.....	
3.7.3 Verify Axis Function.....	
3.7.4 Installation Troubleshooting	
3.8 Controller Customization.....	
4. Operation.....	36
4.1 Control Locations.....	36
4.1.1 Operator Box.....	36
4.1.2 PathPilot Interface.....	36
4.1.3 Accessory Port.....	37
4.2 Initializing the Mill.....	37
4.2.1 Vital Reference.....	37
4.3 Jogging.....	37
4.3.1 Keyboard.....	37
4.3.2 Jog Shuttle.....	38
4.4 Spindle Controls.....	38
4.4.1 Spindle Control Via PathPilot Interface.....	38

PREFACE

4.4.2	Changing Spindle Speed Range.....	39
4.5	Tool Holders.....	40
4.5.1	Install Drawbar and TTS Collet.....	40
4.5.2	Change TTS Tool Holder.....	41
4.5.3	Tips on Using TTS.....	41
4.6	Part Setup/Workholding.....	42
5.	Intro to PathPilot.....	43
5.1	Making Your First Part.....	43
5.1.1	Reference the Mill.....	43
5.1.2	Prepare the Workpiece.....	44
5.1.3	Prepare the Tools.....	44
5.1.4	Understand Mill Position, Work Offsets and Tool Offsets.....	44
5.1.5	Set the Length Units.....	45
5.1.5.1	Programming in Inches.....	45
5.1.5.2	Programming in Millimeters.....	45
5.1.6	Touch Off the Workpiece to Set Work Offsets.....	46
5.1.7	Touch Off the Workpiece to Set Tool Length Offsets.....	47
5.1.8	Write the G-code.....	48
5.1.8.1	Operation 1.....	49
5.1.8.2	Operation 2.....	51
6.	PathPilot Interface.....	53
6.1	Overall Layout.....	53
6.2	Persistent Controls.....	54
6.2.1	Program Control Group.....	54
6.2.2	Position Status Group.....	56
6.2.3	Manual Control Group.....	57
6.3	Keyboard Shortcuts.....	60

6.4 Main Tab.....	61
6.4.1 Selecting a Recent G-code Program File.....	61
6.4.2 Working in the G-code Window.....	61
6.4.2.1 Setting a New Start Line.....	62
6.4.2.2 Expanding the G-code Window.....	62
6.4.3 Manually Entering Commands.....	62
6.4.3.1 Searching in the Code.....	62
6.4.4 Working in the Tool Path Window.....	64
6.4.4.1 Changing the View of the Tool Path Window.....	64
6.5 File Tab.....	64
6.5.1 Managing Files.....	64
6.5.1.1 Transferring Files or Folders from a USB Drive.....	65
6.5.2 Loading G-code.....	65
6.5.3 Editing G-code.....	66
6.5.3.1 Editing G-code with a Text Editor.....	66
6.5.3.2 Editing G-code with Conversational Programming.....	66
6.6 Settings Tab.....	67
6.6.1 Selecting the Tool Changer Type.....	67
6.6.2 Changing the Network Name.....	68
6.6.3 Disabling Reference Switches.....	68
6.6.4 Limiting a G30/M998 Move.....	69
6.6.5 Enabling Feeds and Speeds Suggestions in Conversational Programming.....	69
6.6.6 Enabling Accessories.....	69
6.6.6.1 Enabling CNC Scanner.....	69
6.6.6.2 Enabling Soft Keyboard.....	69
6.6.6.3 Enabling USB I/O Board.....	70
6.7 Offsets Tab.....	70

PREFACE

6.7.1 Tool Tab.....	70
6.7.1.1 Tool Measuring Techniques.....	71
6.7.1.2 Creating Tool Descriptions.....	72
6.7.2 Offsets Table.....	75
6.7.3 Tool Offset and Fixture Information Backup.....	75
6.7.4 Work Tab.....	76
6.8 Conversational Tab.....	76
6.8.1 Using Feeds and Speeds Suggestions.....	76
6.8.1.1 Adjusting DRO Values.....	78
6.8.1.2 Resetting DRO Values.....	78
6.8.1.3 Using Chip Load Information.....	78
6.8.2 Face Tab.....	79
6.8.3 Profile Tab.....	80
6.8.4 Pocket Tab.....	82
6.8.4.1 Rectangular.....	82
6.8.4.2 Circular.....	83
6.8.5 Drill Tab.....	85
6.8.6 Thread Mill Tab.....	87
6.8.7 Engrave Tab.....	89
6.8.8 DXF Tab.....	91
6.8.8.1 Working with Layers and Shapes.....	91
6.8.8.2 Working in the Preview Window.....	92
6.9 Probe Tab.....	92
6.9.1 XYZ Probe Tab.....	93
6.9.2 Rect/Circ Tab.....	94
6.9.3 Probe/ETS Setup Tab.....	95
6.10 ADMIN Commands.....	95

7. Programming.....	96
7.1 Definitions.....	96
7.2 G-code Programming Language.....	99
7.2.1 Overview.....	99
7.2.2 Block.....	99
7.2.3 Real Value.....	99
7.2.4 Number.....	99
7.2.5 Formatting G-code Blocks	100
7.2.6 Optional Program Stop Control – (M01 BREAK).....	103
7.2.7 Additional G-code Formatting Notes.....	104
7.2.7.1 Repeated Items.....	104
7.2.7.2 Order of Execution	104
7.2.7.3 Error Handling	106
7.2.7.4 Modality and Modal Commands	106
7.2.7.5 Modal Groups	106
7.2.7.6 Default Modes.....	107
7.3 G-codes	107
7.3.1 Rapid Linear Motion – G00	109
7.3.2 Linear Motion at Feed Rate – G01	110
7.3.3 Arc at Feed Rate – G02, G03.....	111
7.3.3.1 Radius Format Arc	111
7.3.3.2 Center Format Arc	112
7.3.4 Dwell – G04	115
7.3.5 Set Offsets – G10.....	115
7.3.5.1 Set Tool Table – G10 L1.....	115
7.3.5.2 Set Coordinate System – G10 L2.....	115
7.3.5.3 Set Tool Table – G10 L10.....	116

PREFACE

7.3.5.4 Set Tool Table – G10 L11.....	117
7.3.5.5 Set Coordinate System – G10 L20.....	117
7.3.6 Plane Selection – G17, G18, and G19	118
7.3.7 Length Units – G20, G21	118
7.3.8 Return to Pre-defined Position – G28, G28.1	118
7.3.9 Return to Pre-defined Position – G30, G30.1	119
7.3.10 Straight Probe – G38.x	119
7.3.10.1 Using the Straight Probe Command.....	120
7.3.11 Cutter Compensation – G40, G41, and G42.....	121
7.3.11.1 Cutter Compensation Off – G40.....	121
7.3.11.2 Cutter Compensation On – G41, G42.....	122
7.3.12 Dynamic Cutter Compensation – G41.1, G42.1.....	122
7.3.13 Apply Tool Length Offset – G43.....	123
7.3.14 Engrave Sequential Serial Number – G47.....	124
7.3.15 G49 Cancel Tool Length Compensation – G49.....	124
7.3.16 Absolute Coordinates – G53	125
7.3.17 Select Work Offset Coordinate System – G54 to G59.3.....	125
7.3.18 Set Exact Path Control Mode – G61	125
7.3.19 Set Blended Path Control Mode – G64	126
7.3.20 Distance Mode – G90, G91	126
7.3.21 Arc Distance Mode – G90.1, G91.1.....	126
7.3.22 Temporary Work Offsets – G92, G92.1, G92.2, and G92.3	127
7.3.23 Feed Rate Mode – G93, G94, and G95.....	127
7.3.24 Spindle Control Mode – G96, G97.....	128
7.4 Canned Cycles.....	129
7.4.1 High Speed Peck Drilling Cycle – G73.....	131
7.4.2 Cancel Active Canned Cycle – G80.....	131

7.4.3	Simple Drilling Cycle – G81.....	132
7.4.4	Simple Drilling Cycle (dwell) – G82.....	134
7.4.5	Peck Drilling Cycle – G83.....	134
7.4.6	Boring Cycle (feedrate out) – G85.....	134
7.4.7	Boring Cycle (dwell, rapid out) – G86.....	135
7.4.8	Boring Cycle (dwell, manual out) – G88.....	135
7.4.9	Boring Cycle (dwell, feedrate out) – G89.....	135
7.5	Built-in M-codes	136
7.5.1	Program Stop and Program End – M00, M01, M02, and M30	136
7.5.2	Spindle Control – M03, M05	137
7.5.3	Tool Change – M06.....	137
7.5.4	Coolant Control – M07, M08, and M09	137
7.5.5	Override Control – M48, M49	138
7.5.6	Feed Override Control – M50.....	138
7.5.7	Spindle Speed Override Control – M51.....	138
7.5.8	Set Current Tool Number – M61.....	138
7.5.9	Set Output State – M64, M65.....	139
7.5.10	Wait on Input – M66.....	140
7.6	Other Input Codes	140
7.6.1	Feed Rate – F	140
7.6.2	Spindle Speed – S	140
7.6.3	Change Tool Number – T.....	141
7.7	Advanced Programming with Parameters and Expressions.....	141
7.7.1	Parameters	141
7.7.2	Parameter Types.....	143
7.7.2.1	Numbered Parameters.....	143
7.7.2.2	Subroutine Parameters.....	145

PREFACE

7.7.2.3	Named Parameters.....	145
7.7.3	Expressions.....	145
7.7.3.1	Binary Operators.....	145
7.7.3.2	Functions.....	147
7.8	Programming with Subroutines.....	147
7.8.1	Subroutine Labels and Subroutine Keywords.....	147
7.8.1.1	Defining a Subroutine.....	148
7.8.1.2	Calling a Subroutine.....	149
7.8.1.3	Conditional Subroutines.....	150
7.8.1.4	Repeating Subroutines.....	150
7.8.1.5	Looping Subroutines.....	151
8.	Accessories.....	153
8.1	Stands and Enclosures.....	153
8.1.1	Full Enclosure Kit.....	153
8.1.2	Stand.....	153
8.2	Coolant System Options.....	153
8.2.1	Flood Coolant Kit.....	153
8.2.2	Spray Coolant.....	154
8.2.3	Coolant Hose and Accessories.....	154
8.2.4	Tramp Oil Pillow.....	154
8.3	Mill Options.....	154
8.3.1	LED Spindle Light.....	154
8.3.2	External Contactor Kit.....	154
8.3.3	USB M-code I/O Interface Kit.....	155
8.3.4	Kress Spindle Kit.....	155
8.3.5	Probe.....	155
8.3.5.1	Calibrating Probe Tip.....	156

8.3.5.2 Measuring Probe Tip Diameter.....	157
9. Maintenance.....	158
9.1 Regular Maintenance.....	158
9.1.1 Rust Prevention.....	159
9.1.2 Way Covers.....	159
9.1.3 Flood Coolant System.....	159
9.1.4 Lubrication System.....	159
9.1.4.1 Manual Oiler Specifics.....	160
9.1.5 Drawbar and TTS Collet.....	161
9.2 Spindle Belt.....	161
9.3 Advanced Maintenance.....	161
9.3.1 Overview.....	161
9.3.2 Definitions.....	162
9.3.2.1 How to Measure Lost Motion.....	162
9.3.3 Gib Adjustment.....	164
9.3.3.1 Overview.....	164
9.3.3.2 Adjustment Procedure.....	164
9.3.4 Angular Contact Bearing Preload Adjustment.....	165
9.3.4.1 Overview.....	165
9.3.4.2 Adjustment Procedure.....	166
9.3.5 Geometry Adjustment of Precision Mating Surfaces.....	167
9.4 Spindle Bearing Adjustment.....	167
9.5 Transporting the Mill.....	167
10. Diagrams and Parts List.....	169
10.1 Spindle Head Assembly (exploded view).....	169
10.2 Z-axis Column Assembly (exploded view).....	171
10.3 X-Y Axis Assembly (exploded view).....	173

PREFACE

10.4	Electrical Cabinet.....	176
10.5	Connections.....	178
10.6	Stepper Connections.....	179
10.7	Lubrication System.....	180

1. OVERVIEW

Tormach PCNC mills are intended for use as general purpose CNC mills. Pictured below is a typical PCNC 440 mill set up, including several options.



Figure 1.1

Item #	Component	Item #	Component
1	Electrical Cabinet	6	Spindle Door
2	Main Disconnect Switch	7	Spindle Door Latch
3	DB-25 Connection and Accessory Input	8	Chip Pan
4	Serial Number Plate	9	Deluxe Stand (optional)
5	Controller Compartment		

OVERVIEW

1.1 Specifications

Mechanical		
Key Dimensions (machine table)	Length	18"
	Width	6.25"
	T-Slot Width	10 mm
	T-Slot Center-to-Center Distance	50 mm
	Number of Standard T-Slots	Three along X-axis
	Maximum Weight on Table	150 lbs.
	Spindle Nose to Table (~max)	12"
	Spindle Nose to Table (~min)	2"
	Spindle Center to Z-axis Way Cover	6.5"
Travels	X-axis	10"
	Y-axis	6.25"
	Z-axis	10"
Spindle	Speed Range	300-10,000 RPM
	Maximum Rating	3/4 hp (600 W)
	Drive System	Belt Driven (two positions) Low Belt: 300-4500 RPM High Belt: 650-10,000 RPM
	Taper	R8
Feed Rates	Rapids on X	135 IPM
	Rapids on Y	135 IPM
	Rapids on Z	110 IPM
	Max Cutting	135 IPM (X, Y), 110 IPM (Z)
Temperature	Operating Range	45°F-100°F (7°C-38°C)
Electrical		
Power Requirements		115 VAC
Recommended Circuit Amperage ¹		15 AMP

¹Dedicated circuit recommended; do not use with a ground-fault interrupter (GFI).

SITE PLANNING AND PREP

2. SITE PLANNING AND PREP

This section covers required site preparations prior to placing PCNC mill in service.

2.1 General Site Requirements

The area should be well lit, dry, have proper ventilation, provide for unobstructed machine motion/operation, and ensure unrestricted access to PCNC mill controls.

2.1.1 Space Requirements

Minimum floor space requirements are as follows:

Width	Depth	Height
40"	24"	44"

NOTE: Allocate additional space to allow access to rear of mill for maintenance or repairs.

2.2 Electrical Requirements



WARNING! Electrical Shock Hazard: Electrical connections must be performed by a certified electrician. Failure to do so may result in death and/or serious injury.

Primary	Recommended Circuit Amperage
115 VAC, 50/60 Hz	15 A

2.2.1 Grounding

All power inputs to the PCNC 440 must be properly grounded. Check continuity between bare metal on mill frame and true earth ground (water pipe or similar) to ensure proper grounding.

2.2.2 Plug Pattern

The PCNC 440 is shipped with a 3-prong plug. This plug is designed to be used with a 3-prong outlet.

2.2.3 Ground Fault Interrupter (GFI) Use

Primary power for the PCNC 440 should not be protected by a ground fault interrupter (GFI), as this interferes with the proper operation of the PCNC mill's spindle controller.

SITE PLANNING AND PREP

2.2.4 Electrical Noise

Primary power should be provided by a dedicated circuit. At the minimum, it should be isolated from electrically-noisy devices. In particular, high-inductive loads from vacuum cleaners, air compressors, etc., can be troublesome and the source of controller malfunction.

At sites where this is not possible, a double-conversion uninterruptible power supply (UPS) should be considered for 115 VAC circuits.

2.2.5 Options for Electrically Non-conforming Sites

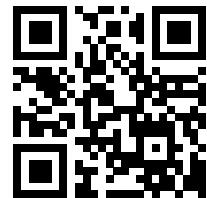
The following device may be considered for sites that do not conform to the electrical requirements detailed in this section. Consult with an electrician to determine suitability for the specific site.

- Step-up/Step-down Transformer (PN 32009): used to reduce 230 VAC line voltage to 115 VAC, as required by the PCNC 440; commonly used for PCNC 440 mills located outside of the USA and Canada.

3. INSTALLATION

This chapter covers basic installation of a PCNC 440 mill, which takes approximately two hours. This estimate does not include optional accessories like enclosures, power drawbars, or automatic tool changers (ATC).

Scan the QR code (at right) to view a list of related technical documentation, operator manuals, and support videos: torma.ch/install



Required for set up:

- Power Strip

Recommended for installation/set up:

- | | |
|--|---|
| <ul style="list-style-type: none"> • Gloves • Eye Protection • Pry Bar • Pallet Jack | <ul style="list-style-type: none"> • Engine Hoist • Socket Set • Metric Wrench Set |
|--|---|

3.1 Receiving, Uncrating, and Initial Inspection



WARNING! Transport and Lift Hazard: The transport, lifting, and moving of mill should be done by qualified professionals. Failure to do so could result in death, serious injury, and/or mill damage.

3.1.1 Shipment Arrival

Depending on products and options ordered, the PCNC system arrives in two or more shipments:

- PCNC mill (freight)
- Chip pan (freight; shipped separate from mill)
- Stand (freight)
- Accessory shipment (freight and/or parcel service)

IMPORTANT! Specific shipping information is displayed on packing list. Wait until all shipments are received before beginning installation.

3.1.2 Moving the Crate

The crated mill is shipped on a pallet and can be off-loaded from a truck with a tailgate lift and moved (on smooth surfaces) using a hydraulic pallet jack to the installation location (see **Figure 3.1**).



Figure 3.1

INSTALLATION

3.1.3 Initial Uncrating

 **CAUTION! Sharp Objects:** Be sure to wear gloves when uncrating mill. Failure to do so could result in serious injury.

Using a pry bar, open and disassemble shipping crate by removing the top of crate first, followed by four sides. Be careful with nails. **Figure 3.2** shows the mill with the shipping crate removed.

3.1.4 Shipping Damage or Shortages

Once received, inspect and note any shipping damage that may have occurred during transit. Check received goods against packing list; any damage claims or shortages must be addressed within 30 days of receipt.

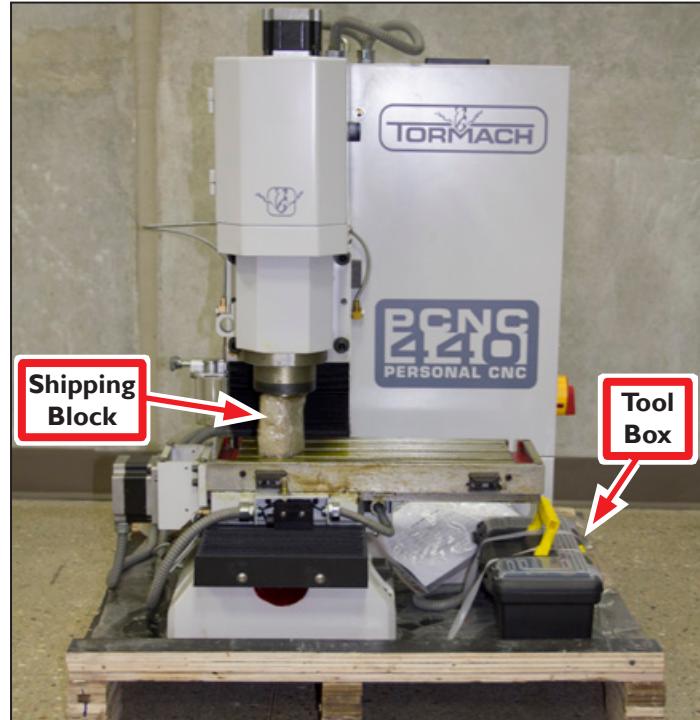


Figure 3.2

3.2 Installation Sequence

If the mill was purchased with additional accessories or optional kits, the following installation sequence is recommended:

1. Basic installation (see *Basic Installation Procedure* later in this chapter)
2. Installation validation (see *Validate Basic Installation* later in this chapter)
3. Power Drawbar
4. Automatic Tool Changer (ATC)
5. Full Enclosure

NOTE: For items 3-5, refer to documentation that ships with the product.

3.3 Basic Installation Procedure

Follow the steps in this section to complete basic mill installation.

3.3.1 Stand Assembly

If purchased, the PCNC 440 Stand (PN 35671) should be assembled first. Refer to documentation that ships with the stand for information on assembly.

3.3.2 Remove Pallet Items

- Carefully cut straps to remove plastic Tool Box strapped to pallet (see **Figure 3.2**); set aside for use later. This box contains tools required for installation.

NOTE: The mill's drawbar is located in the Tool Box. For information on drawbar installation, refer to chapter 4, Operation.

- Remove X-axis Motor Cover Plate and stainless steel X-axis Limit Switch Cover Plate (see **Figure 3.3**); set aside for installation later.

IMPORTANT! Cut straps securing operator box to pallet and keep together with mill for lifting.

3.3.3 Lift and Move Mill



WARNING! Transport and Lift Hazard:

The transport, lifting, and moving of mill should be done by qualified professionals. Failure to do so could result in death, serious injury, and/or mill damage.

3.3.3.1 Remove Mill from Pallet

The mill is secured to the shipping pallet with four bolts/nuts. Before lifting mill, use 16 mm wrench to remove nuts connecting mill to pallet and discard.

3.3.3.2 Lift Mill

Lift the mill from above with either a forklift or an engine hoist. The single Eye Bolt on top of the column is suitable for lifting the entire weight of the mill (see **Figure 3.4** and inset). Do not attempt to lift mill from above using any other method. If using an engine hoist, the minimum distance between hoist legs is 22", the distance necessary to straddle the pallet.

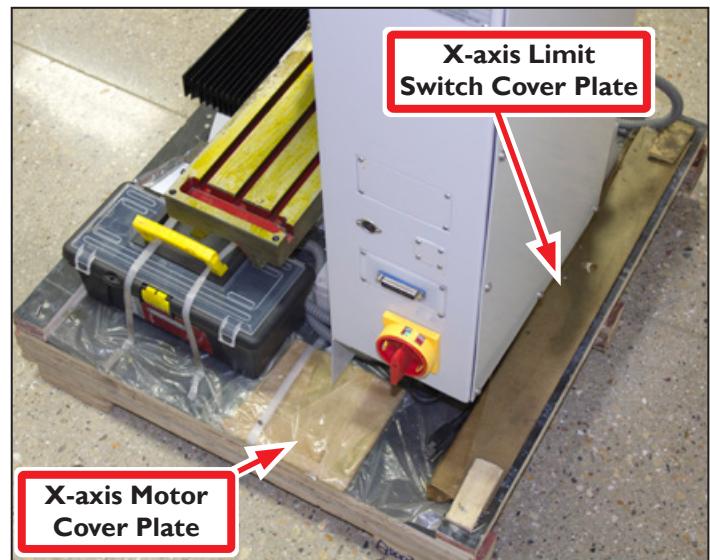


Figure 3.3



Figure 3.4

INSTALLATION

3.3.4 Attach Mill to Stand



WARNING! Crush Hazard: Keep hands and body parts clear when lowering mill onto stand. Failure to do so could result in death and/or serious injury.

1. Place the optional chip pan (PN 35673) on top of optional stand (PN 35671).
2. Lower mill slowly while lining up holes in chip pan, mill, and stand (see **Figure 3.5**).
3. Using four M12 x 100 mm studs, washers, and nuts, attach chip pan, mill, and stand together; tighten nuts securely.

3.3.4.1 Non-Tormach Stands

If an alternative stand is used, ensure:

- Stand has a minimum weight capacity of 1000 lbs
- Mill is securely connected to stand
- Stand is stable and secure

3.3.5 X-axis Motor Cover Plate

Install the X-axis Motor Cover Plate using provided screws to attach it to left side of machine table (see **Figure 3.6**).

3.3.6 X-axis Limit Switch Cover Plate

Install the X-axis Limit Switch Cover Plate using provided screws to attach it to front of machine table (see **Figure 3.6**).

3.3.7 Manual Oiler

Retract and release plunger until oil moves through system. After that, pull plunger each time mill is powered on and after every four hours of operation. For more information on providing oil to the machine, refer to chapter 9, *Maintenance*.

3.3.8 Flood Coolant System

Fill the reservoir with pre-mixed coolant; refer to dilution instructions for coolant product. For information on installation and use of the optional Flood Coolant Kit (PN 35719), refer to documentation that ships with the product.

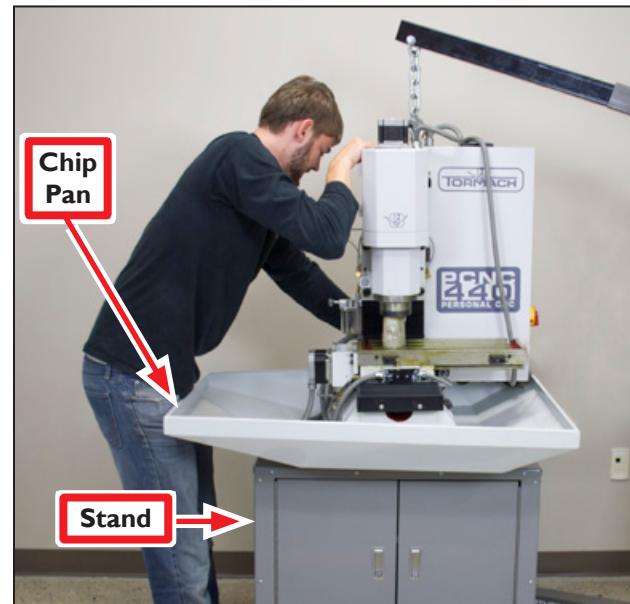


Figure 3.5

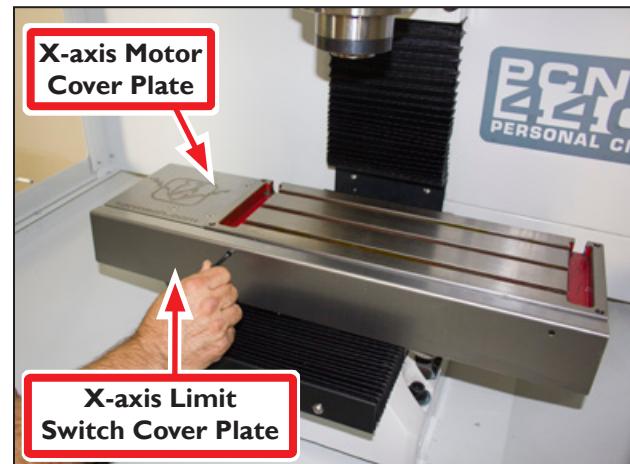


Figure 3.6

3.3.9 Install PathPilot Controller

Review the connections on the front and rear of the PathPilot® controller as shown in **Figure 3.7** and **Figure 3.8**. When all connections are complete, place controller in a location away from dust, debris, and any liquids.

PathPilot Controller (PN 35286)

Item #	Connection or Component
1	Power/Reset with LED
2	USB Connectors
3	Optical Drive
4	Hard Drive LED
5	PS/2 Connector
6	USB Connectors
7	DVI Connector
8	VGA Connector
9	DP*
10	HDMI*
11	Blue USB Connectors (4)
12	Ethernet Connector
13	Audio Connections*
14	Mill Interface Port
15	Voltage Setting Switch
16	PCI Expansion Slot
17	AC Power Connector

*Do not use these controller features.

Set up and connect the PathPilot controller as follows:

1. Confirm Voltage Setting Switch (#15) is set to proper voltage for geographic location.
2. Plug the power cord into the AC Power Connector (#17) on the PathPilot controller. Plug the other end into a power strip (not included).
3. Plug DB-25 interface cable into the DB-25 outlet on side of mill (see **Figure 3.12**).
4. Connect monitor to either the DVI Connector (#7) or the VGA Connector (#8).
5. Connect loose end of DB-25 interface cable to Mill Interface Port (#14).
6. Connect jog shuttle (optional), keyboard, ATC (optional), and USB I/O board (optional) to Blue USB Connectors (#11).
7. Connect other USB devices to USB Connectors (#2 or #6); these are not for use with wireless keyboard/mouse.



Figure 3.7

INSTALLATION

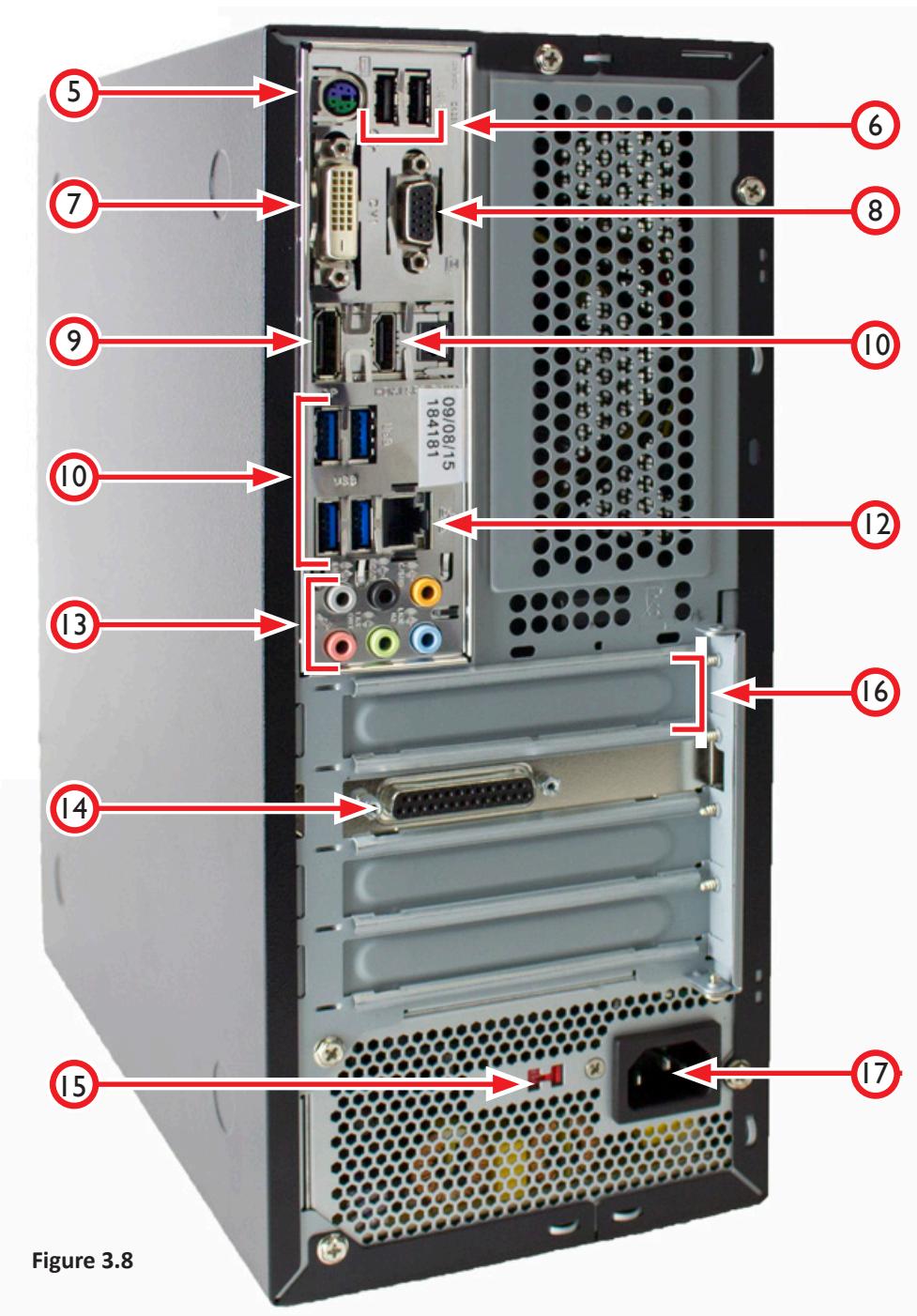


Figure 3.8

3.4 Essential Controls Overview

Check to ensure your local power supply meets the requirements detailed in chapter 2, *Site Planning and Prep.*

3.4.1 E-stop, Start, Reset, and Main Disconnect

NOTE: Before continuing, review Power Off/On Procedure later in this chapter.

E-stop (emergency stop)

The mill has one emergency stop button or E-stop pre-installed on the Operator Box (see **Figure 3.9**). The E-stop terminates all motion and spindle function. Depress the red E-stop and it locks in the Power Off position (see **Figure 3.10**). Turn the red E-stop clockwise a quarter-turn to release it; press the green Start button to power back on (see **Figure 3.9**).

Start

The mill's green Start button, located on the Operator Box (see **Figure 3.9**), powers on circuits for the axis drives. The PathPilot interface indicates the mill is powered on and ready to operate when the *Machine OK* LED is illuminated (see **Figure 3.13**).

NOTE: Once E-stop is depressed, Start button is inoperative until the E-stop is released.

Reset

Click the blinking *Reset* button on the PathPilot interface to establish communications between the controller and the mill (see **Figure 3.13**).

Operator Box

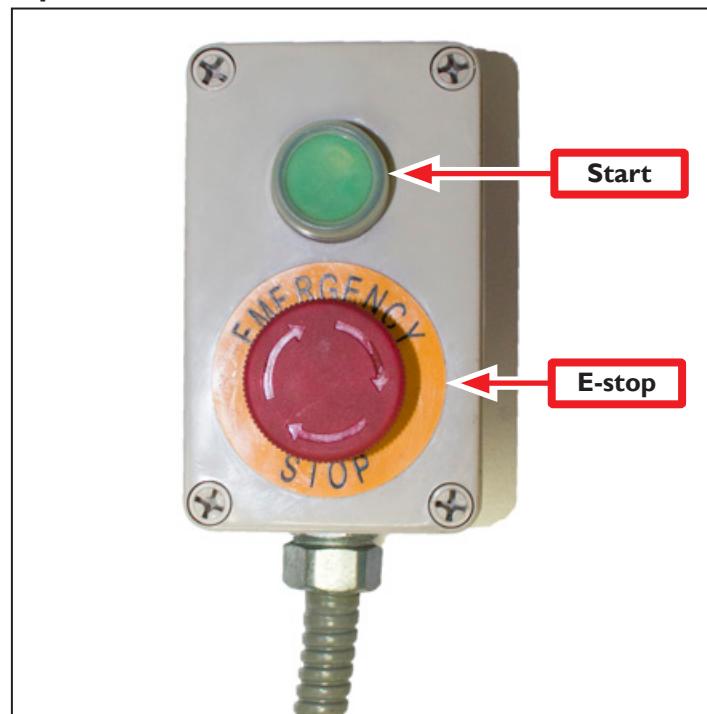


Figure 3.9

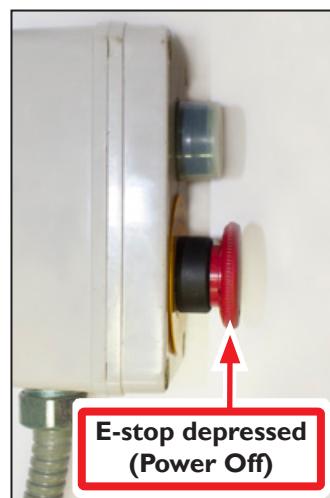


Figure 3.10

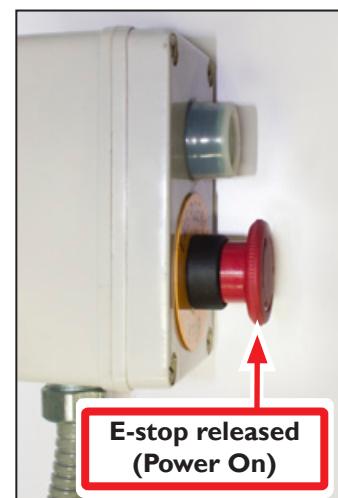


Figure 3.11

INSTALLATION

Main Disconnect

The Main Disconnect switch, located on the right side of the electrical cabinet, is used to power the mill on and off (see **Figure 3.12**). When the Main Disconnect is switched to power *Off*, this disconnects the primary supply power to the mill.

3.5 Power Off/Power On Procedure

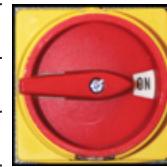
IMPORTANT! Do not power on motors and drives via the green Start button before powering on the controller that oversees their operation. Make sure that the controller is on and the PathPilot interface is loaded before powering on the mill. Likewise, make sure to depress the red E-stop before powering off the controller using the Exit button on the PathPilot interface (see **Figure 3.13**). Confirm mill powers off and on correctly per the Power Off/On Procedure below.



Figure 3.12

Power Off/On Procedure

Power Off	1. Push in red <i>E-stop</i> button
	2. Click <i>Exit</i> on screen; when prompted click <i>OK</i> to power off
	3. Turn PathPilot Controller power strip off
	4. Turn Main Disconnect <i>Off</i> (see image at right)
Power On	1. Turn PathPilot Controller power strip on
	2. After software loads, turn Main Disconnect <i>On</i> (see image at right)
	3. Turn red <i>E-stop</i> button clockwise to release
	4. Press green <i>Start</i> button
	5. Click <i>Reset</i> on screen



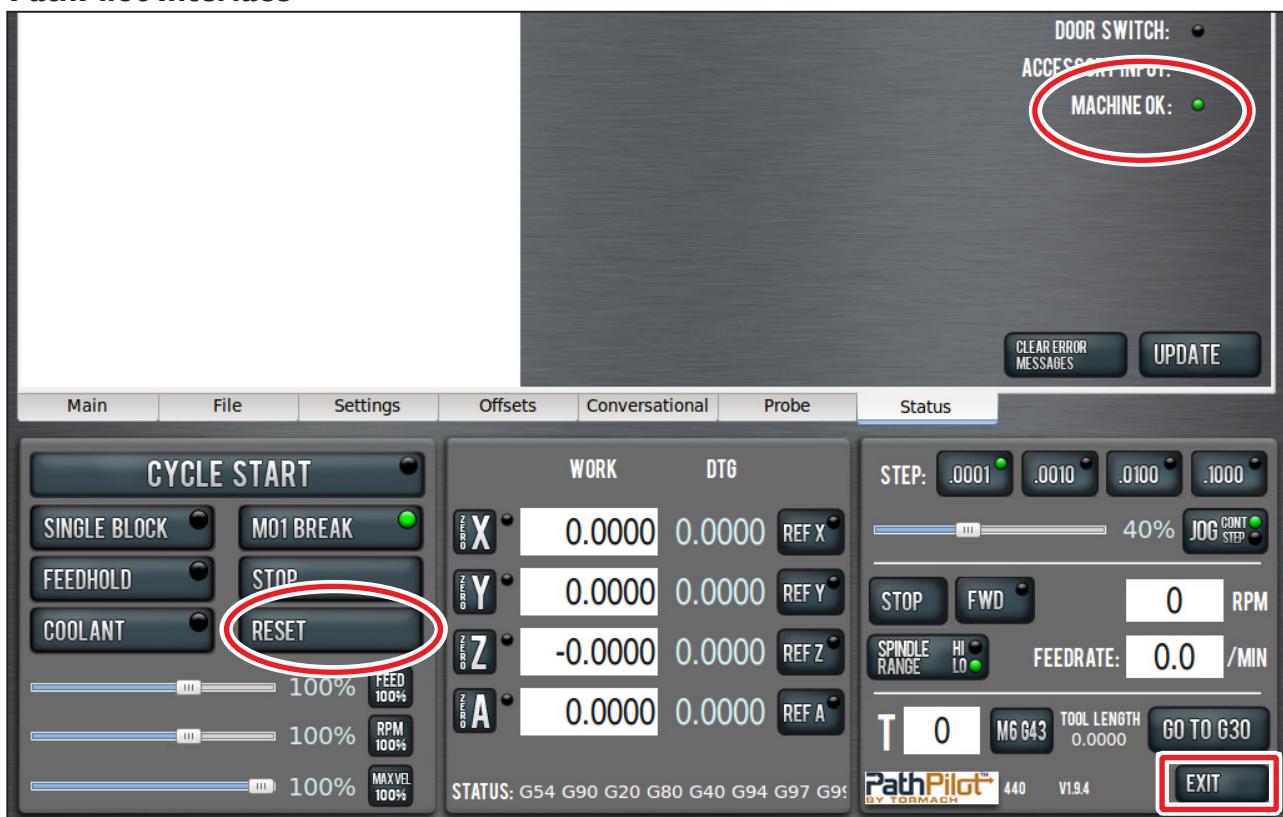
PathPilot Interface

Figure 3.13

3.6 Initial PathPilot Controller Configuration



WARNING! Unattended Operation: Machine is not designed to operate unattended. Do not leave machine unattended during operation. When machine is not in use, turn the main disconnect off. Failure to do so could result in death, serious injury, and/or machine damage.

Power on the PathPilot controller by switching the power strip on (not included); PathPilot controller will boot. Turn the Main Disconnect switch to *On* (see **Figure 3.12**).

The first time the PathPilot controller is powered on it starts a configuration process that allows the operator to configure the PathPilot operating system to the particular machine (PCNC 1100 mill, PCNC 770 mill, PCNC 440 mill, or 15L Slant-PRO lathe). Follow the on-screen instructions to complete controller configuration. After configuration, PathPilot automatically launches; the controller automatically loads PathPilot for the selected machine when the controller is powered on in the future.

INSTALLATION

3.7 Validate Basic Installation

Validate basic setup prior to installing any accessory kits.

IMPORTANT! Follow the Power Off/On Procedure detailed earlier in this chapter. After powering on, jog the Z-axis up to remove shipping block between spindle nose and bed.

3.7.1 Verify Spindle Function

With the controller and mill powered on and the spindle door closed, select the spindle range position (high or low) for the mill's pulley belt on the PathPilot interface (see **Figure 3.14**). For more information, refer to chapter 4, *Operation*. Ensure the spindle is clear of any obstacles and set spindle speed to 3000 RPM (see **Figure 3.14**).

Click the *Fwd* button (see **Figure 3.14**) and the spindle begins rotation. Click the *Stop* button and the spindle stops. For more information on controlling the spindle, refer to chapter 4, *Operation*.

3.7.2 Verify Limit Switch Function

Limit switches prevent the mill from exceeding its travel limits and provide a reference location during the mill homing procedure. There are three limit switches, one for each axis of motion (X, Y, and Z), as shown in **Figure 3.15**. If a limit switch is triggered, the mill is placed in a reset state. Verify function of the limit switches as follows:

1. On the PathPilot interface, click the *Status* tab (see **Figure 3.16**).
2. Manually depress X, Y, and Z limit switches by hand (see **Figure 3.15**).
3. Verify that the corresponding LED light illuminates on the *Status* screen (see **Figure 3.16**).
4. After verifying limit switch function, click the flashing *Reset* button (see **Figure 3.16**).

Limit Switches



Figure 3.15



Figure 3.14

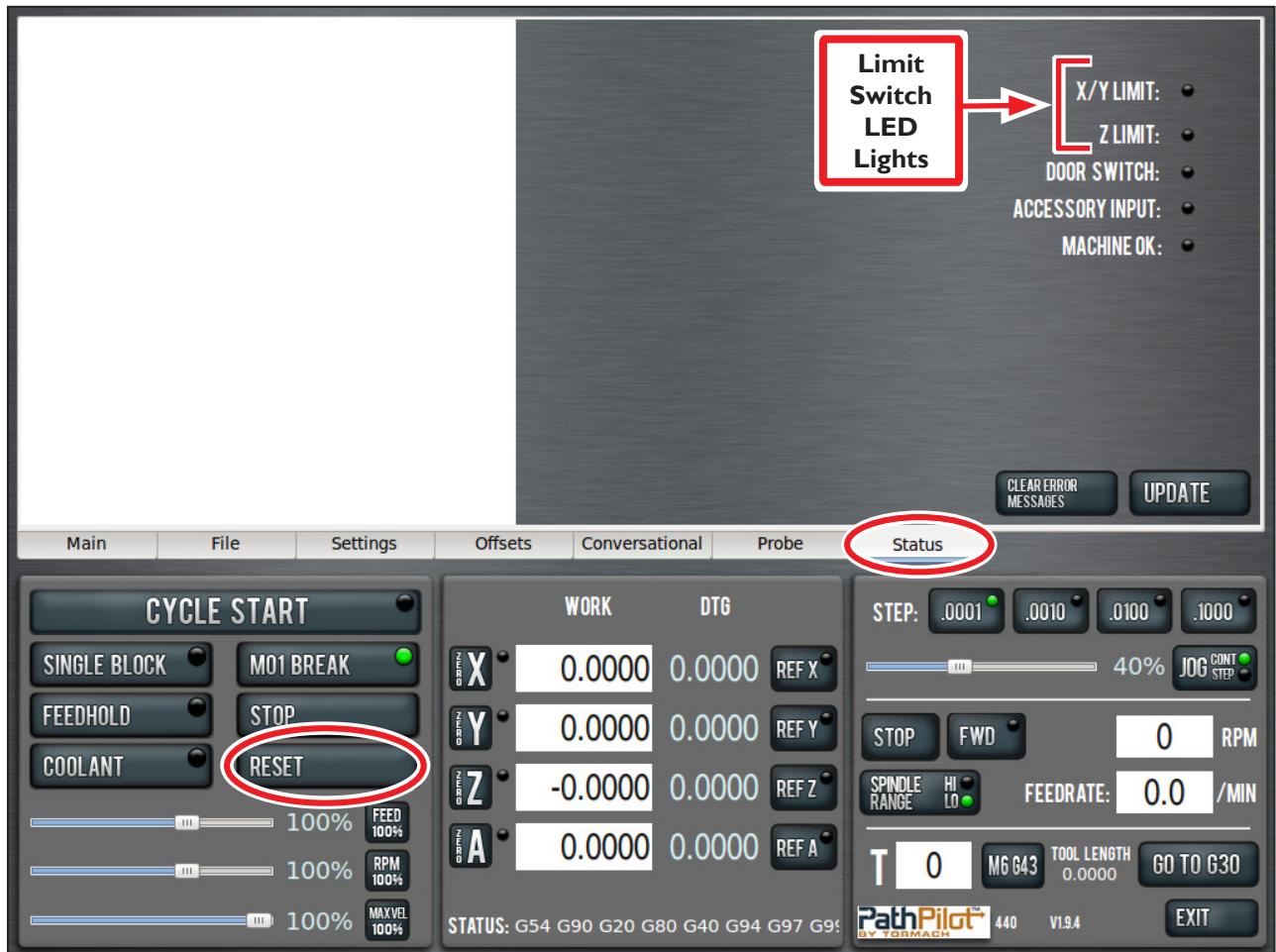


Figure 3.16

3.7.3 Verify Axis Function

1. Reference the mill by clicking the *Ref Z*, *Ref X*, and *Ref Y* buttons (see **Figure 3.17**); the mill moves.
2. Next, switch to the *Main* screen.
3. Use keyboard to verify axis motion:
 - a. To move the X-axis, use the \leftarrow/\rightarrow arrow keys.
 - b. To move the Y-axis, use the \uparrow/\downarrow arrow keys.
 - c. To move the Z-axis, use the *Page Up/Page Down* keys.

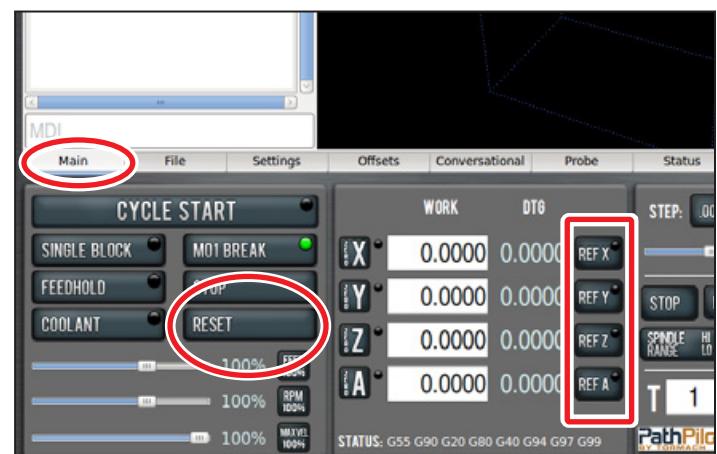


Figure 3.17

INSTALLATION

4. To test the (optional) jog shuttle:
 - a. Press the corresponding axis button (X, Y, or Z) to select the axis.
 - b. Twist shuttle ring of jog shuttle to move axis; twist in opposite direction to reverse direction.

3.7.4 Installation Troubleshooting

Upon initial installation, the most likely reason for non-functioning controls is wires that have become loose during transport. Check to ensure all wires inside the electrical cabinet are properly connected.



WARNING! Electrical Shock Hazard: Be sure to power off machine before making any electrical modifications. Failure to do so could result in death and/or serious injury.

1. Power off the mill according to the *Power Off/On Procedure* detailed earlier in this chapter.
2. Using two fingers, firmly tug each wire connection near its termination point. Any loose wires should be re-seated and re-tightened.

3.8 Controller Customization

Date and Time

To set or edit controller's date and time, type **ADMIN DATE** in the MDI field and click *Enter* on keyboard (see **Figure 3.18**). This opens a dialog box to enter or edit date, time, and time zone. Click *Close* when finished.

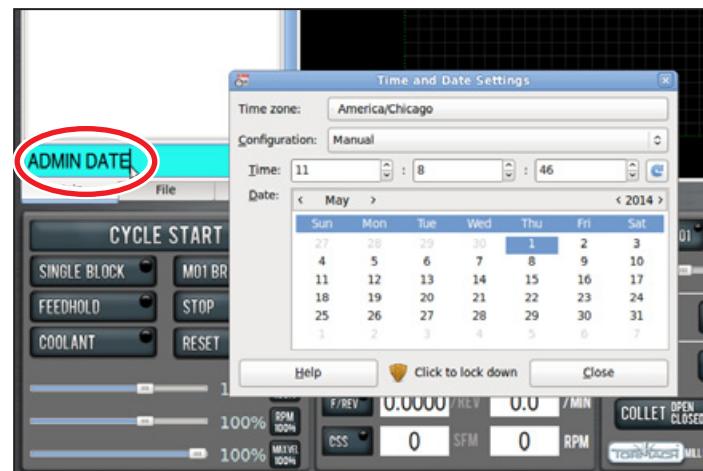


Figure 3.18

Keyboard Language

If you do not have a USA keyboard (QWERTY) layout, type *ADMIN KEYBOARD* in the MDI field and click *Enter* on the keyboard (see **Figure 3.19**).

Next, click on the *Layouts* tab to change the layout of the keyboard to a different language (see **Figure 3.20**). Select a layout and click *Close* when finished.

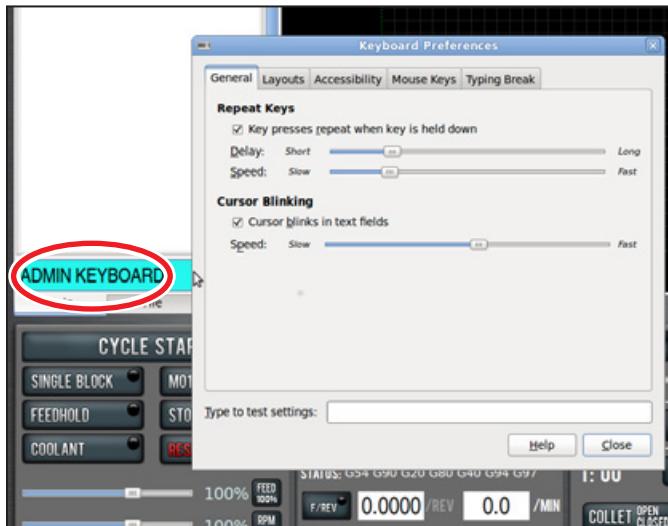


Figure 3.19

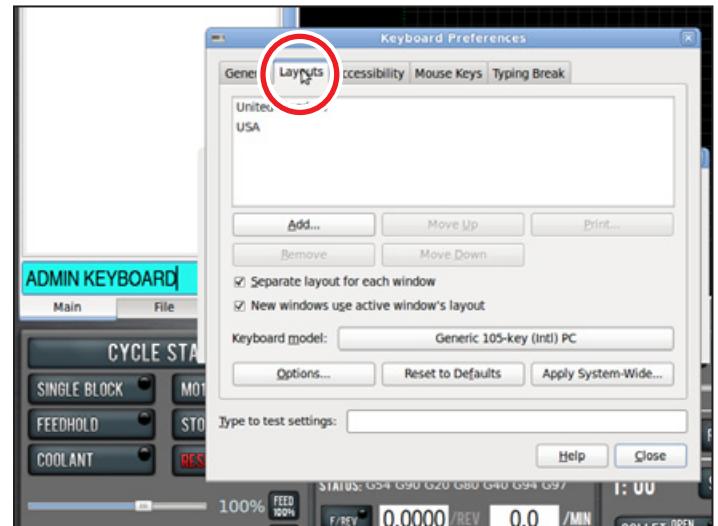


Figure 3.20

Touch Screen (optional)

Refer to documentation that ships with 17" Touch Screen Kit (PN 35575) for information on setup and calibration.

OPERATION

4. OPERATION

This chapter provides an overview of the basic controls of the PCNC 440 mill.



WARNING! Unattended Operation: Machine is not designed to operate unattended. Do not leave machine unattended during operation. When machine is not in use, turn the main disconnect off. Failure to do so could result in death, serious injury, and/or machine damage.

4.1 Control Locations

There are two primary controls: the Operator Box and the (on-screen) PathPilot® interface. All control functions can be found in one or the other location.

4.1.1 Operator Box

The operator box is shown in **Figure 4.1**. It contains physical buttons that control the following functions:

- Start
- E-stop

NOTE: Operator may choose to mount operator box to slots on the right side of the chip pan. Hardware is not included for mounting.

4.1.2 PathPilot Interface

PathPilot is a sophisticated CNC controller for Tormach products. All aspects of the PCNC 440 are controlled via the on-screen PathPilot interface. There are four primary ways for the operator to interact with PathPilot:

- Keyboard
- Mouse
- Jog shuttle (optional)
- Touch screen (optional)

For information on the PathPilot interface, refer to chapter 6, *PathPilot Interface*.

Additional devices can also interface with PathPilot. These include Probe, Tool Setter, CNC Scanner, etc. For information on these accessories, refer to chapter 8, *Accessories*.

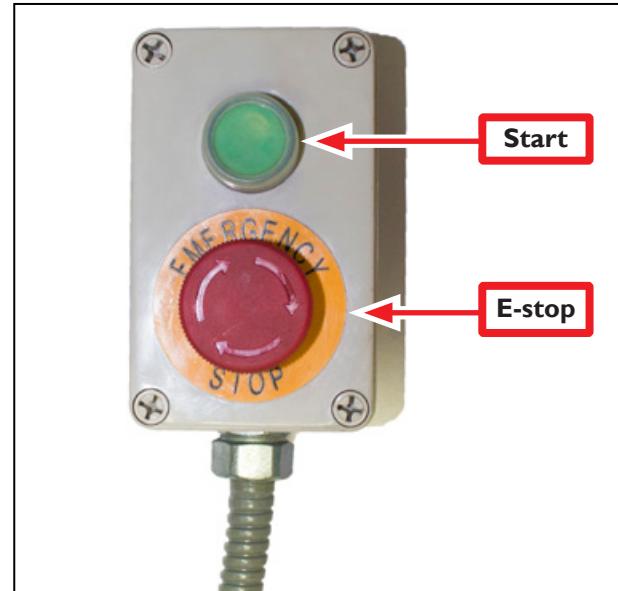


Figure 4.1

4.1.3 Accessory Port

The Accessory Port is located on the side of the electrical panel (see **Figure 4.2**).

4.2 Initializing the Mill

To prepare the mill for motion, the mill must be initialized.

4.2.1 Vital Reference

After powering on, the mill must be referenced in the X-, Y-, and Z-axes. Execute the referencing procedure as follows:

1. Power on the mill following the power off/on procedure detailed in chapter 3, *Installation*.
2. Click the flashing *Reset* button on the PathPilot interface.
3. Click *REF Z*, *REF X*, and *REF Y*.

4.3 Jogging

The mill can be jogged either by using the keyboard (see **Figure 4.3**) or using the optional jog shuttle (see **Figure 4.4**).

4.3.1 Keyboard

- The *right arrow* jogs X-axis in the positive X direction (table moves left of operator).
- The *left arrow* jogs X-axis in the negative X direction (table moves right of operator).
- The *up arrow* jogs Y-axis in the positive Y direction (moves table towards operator).
- The *down arrow* jogs Y-axis in the negative Y direction (moves table away from operator).
- The *Page Up* key jogs the Z-axis in the positive Z direction (moves spindle up).
- The *Page Down* key moves the Z-axis in the negative Z direction (moves spindle down).

NOTE: Jogging is not permitted during G-code program execution or MDI moves.



Figure 4.2

Jogging with Keyboard Keys

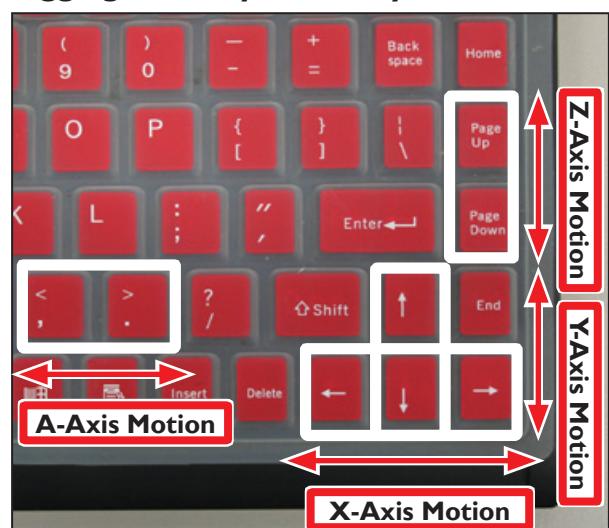


Figure 4.3

OPERATION

Jog Shuttle

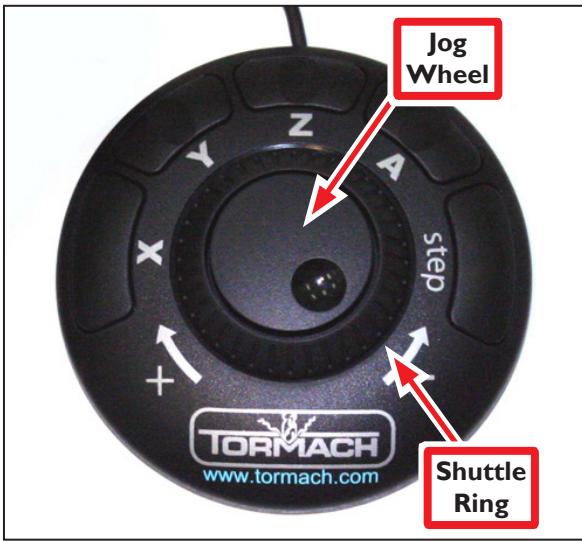


Figure 4.4



Figure 4.5

4.3.2 Jog Shuttle

The Jog Shuttle (PN 30616) is an optional accessory (see **Figure 4.4**) that many operators find increases productivity, especially on short-run jobs requiring extensive setting up of the workpiece and tooling.

The X, Y, Z and A buttons are used to jog the X-, Y-, Z-, and A-axes, respectively. An illuminated LED light beside an axis DRO on the PathPilot interface indicates which axis is selected for jogging. The Step button on the Jog Shuttle cycles through the available jog step sizes. The active size is indicated by an illuminated LED light on the Step Size buttons on the PathPilot interface (see **Figure 4.5**).

For more information on jogging methods, refer to chapter 6, *PathPilot Interface*.

4.4 Spindle Controls

4.4.1 Spindle Control Via PathPilot Interface

Ensure the *Spindle Range* button's LED light (see **Figure 4.6**) correctly corresponds to the spindle belt position, either *Hi* or *Lo*; click to toggle between the two positions. For more information on the procedure to change belt position, refer to *Changing Spindle Speed Range* section later in this chapter.



Figure 4.6

NOTE: A mismatch between the Spindle Range button and actual spindle belt position will result in the commanded speed being different from the indicated RPMs.

To specify spindle RPMs, click the *DRO*. Using the keyboard, type the desired RPM and press *Enter*.

- Click *FWD* to run the spindle clockwise
- Click *Stop* to stop the spindle

4.4.2 Changing Spindle Speed Range

The PCNC 440 mill has two speed ranges:

Low	High
300-4500	650-10,000

The range change is performed by moving the spindle belt from the top pair of pulleys (high speed range) to the lower pair of pulleys (low speed range). To change belt position:



WARNING! Electrical Shock Hazard: Be sure to power off machine before making any electrical modifications. Failure to do so could result in death and/or serious injury.

1. Power off mill according to Power Off/On Procedure detailed in chapter 3, *Installation*.
2. Open spindle door; unlock motor mounting plate using a hex key and pull motor forward. The belt will slacken and can be moved from one pulley to another (see **Figure 4.7**).
3. Re-tighten the belt so that there is approximately 1/8"-1/4" of belt deflection midway between the pulleys with a firm finger press.



Figure 4.7

OPERATION

4.5 Tool Holders

This section describes using tooling compatible with the standard R8 spindle. For more information on other spindle options, refer to the product-specific documentation.

The Tormach Tooling System (TTS®) is the recommended tool holding method for PCNC mills. The advantages of TTS over other tooling options include:

- Exact tool offset repeatability
- Easily adaptable to tool presetting techniques
- Quickest manual tool change time
- Shortest tool change clearance distance
- Compatibility with Tormach power drawbar and Tormach automatic tool changer (ATC)

TTS uses a precision 3/4" collet in combination with a drawbar and interchangeable TTS tool holders. Many different TTS tool holders are available.

4.5.1 Install Drawbar and TTS Collet

1. Identify drawbar and drawbar bushing, located in the included tool box removed from the shipping pallet in chapter 3, *Installation*.
2. Using a clean rag, apply a degreasing agent to the inside taper of the spindle and the entire surface of TTS collet. Wipe clean and dry.
3. Apply a small amount of Anti-seize (PN 31273) grease to:
 - Drawbar threads
 - Outside taper of TTS collet

NOTE: Do not apply grease to the inside surface of the TTS collet.

- Inside taper of R8 spindle
4. Open the spindle door.
 5. Swing the spindle lock so it engages with the flats on the top of the spindle. If necessary, turn the spindle by hand until the flats line up.
 6. Place drawbar bushing on top of the spindle.
 7. Using one hand, insert the TTS collet into the bottom of the spindle.
 8. Twist the collet while applying light upward pressure until the collet groove aligns with the spindle alignment pin; the collet is pushed completely inside the spindle taper.
 9. Using your other hand, insert the drawbar into the drawbar bushing installed in Step 6; rotate the drawbar to engage several threads in the TTS collet.
 10. Place and hold a TTS tool holder inside the TTS collet and against the spindle nose. Using a 13 mm wrench, rotate the drawbar and tighten the collet against the TTS tool holder.

4.5.2 Change TTS Tool Holder

1. Loosen the drawbar by one turn with a 13 mm wrench.
2. Grasp the tool holder in one hand and use a mallet to gently strike the top of the drawbar.
3. Remove the tool holder.
4. Using one hand, insert the TTS collet into the bottom of the spindle.
5. Twist the collet while applying light upward pressure until the collet groove aligns with the spindle alignment pin; the collet is pushed completely inside the spindle taper.
6. Using your other hand, insert the drawbar into the drawbar bushing installed in Step 6; rotate the drawbar to engage several threads in the TTS collet.
7. Place and hold a TTS tool holder inside the TTS collet and against the spindle nose. Using a 13 mm wrench, rotate the drawbar and tighten the collet against the TTS tool holder.

4.5.3 Tips on Using TTS

- Never tighten the TTS collet without a tool holder inserted
- Never change tools while a tool holder is in the spindle as this may damage the spindle alignment pin
- To minimize tool pull-out, periodically wipe tool holder shanks clean and dry with a degreasing agent
- When not in use, apply a protective spray (WD-40® or similar) to prevent surface rust on bare metal surfaces of tool holders
- The drawbar, drawbar bushing, and TTS collet are wear items. Inspect threads and mating surfaces regularly and replace if damage or wear is apparent as it reduces the ability to tighten tools properly

R8 collets and R8 taper tool holders are also compatible. These are installed in a similar manner to the TTS collet as described above, but must be removed completely during each tool change.

NOTE: The PCNC 440 ships with a drawbar with 7/16"-20 UNF thread.

OPERATION

4.6 Part Setup/Workholding

Work must be secured to the table prior to machining. Each PCNC 440 has three 10 mm T-slots that run parallel to the X-axis. The slots are precision ground to +0.008".

A 4" vise is recommended. Tormach offers the following vise:

PN	Description
35285	4" CNC Vise

For more information on proper setup and use of the 4" CNC Vise, refer to documentation that ships with product.

A number of other ways can be employed for workholding. These include toe clamps, fixture plates, chucks, and vacuum tables.

5. INTRO TO PATHPILOT

5.1 Making Your First Part

This chapter outlines how to make your first part on a Tormach mill. It assumes you have no prior experience running a part program on a CNC mill. Even if you have previous CNC experience, following this tutorial gives you an introduction to the controls of the mill. After reading this chapter, read chapters 6 and 7 for details on the PathPilot® operating system. This chapter is only intended to be an introduction to the PathPilot interface and several basic tasks.



Figure 5.1

The first part program uses two tools – a 3/8" end mill and a 1/8" end mill – to make a shallow circular pocket and engrave the text PCNC in a wood 2" x 4" (see **Figure 5.1**). Two tools are used to give you an introduction to tool changes and the difference between work offsets and tool length offsets. For operators using the optional automatic tool changer (ATC), we recommend using manual tool changes for this first part to keep things simple.

5.1.1 Reference the Mill

Follow the Power Off/On Procedure in chapter 3, *Installation*, to turn the PathPilot controller and the mill on. After clicking the flashing *Reset* button, you can reference the X-, Y-, and Z-axes. You should reference the Z-axis first to help avert a crash as it moves the tooling as far as possible from a workpiece or vise. All three axes can be referenced simultaneously by pressing the *Ref* buttons in rapid succession (see **Figure 5.2**).

Reference the axes before operating the mill to establish soft limits (protect the mill from over travel) and give meaning to work offset values. After referencing the axes, the LEDs on the *Ref X*, *Ref Y*, and *Ref Z* buttons illuminate, indicating that the mill has been referenced. While you can jog the mill before referencing, you should not run parts until the mill has been referenced. If a home or limit switch fails to work, manually reference the mill.

E-stopping the mill de-references the axes; be sure to reference again after an E-stop.

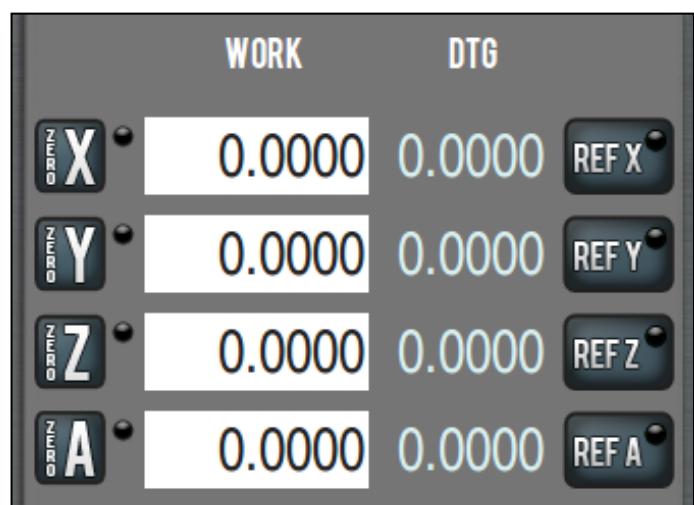


Figure 5.2

INTRO TO PATHPILOT

5.1.2 Prepare the Workpiece

For this introduction to using the mill, use a scrap piece of wood as a workpiece; a 2" x 4" that is at least 4" long will suffice. Using a piece of wood minimizes the chance an end mill is damaged should you get a work or tool offset command wrong while using this tutorial.

5.1.3 Prepare the Tools

For this tutorial you will need a 3/8" diameter end mill to machine the shallow circular pocket and a 1/8" or smaller diameter end mill to engrave the text (see **Figure 5.3**). You will also need a way to hold these tools. Tormach's High Speed Steel End Mill Kit (PN 33465) includes both end mills. You can hold them using TTS-3/8" Set Screw Holders (PN 31820) as shown, or using ER16 or ER20 collet chucks. The 3/8" end mill will be *Tool 1* and the 1/8" end mill *Tool 2* (see **Figure 5.3**).



Figure 5.3

5.1.4 Understand Mill Position, Work Offsets and Tool Offsets

Work Offsets

Work offsets are a concept that allows the operator to think in terms of X/Y/Z coordinates with respect to the part, instead of thinking of them with respect to the mill position – work offsets allow you to assign an origin to any location within the work envelope.

When referencing the mill, it moves to the limit switches and stops at its home position; this is $(X, Y, Z) = (0, 0, 0)$ in mill coordinates – however, these coordinates are not useful to the operator or programmer who wants to think in terms of program coordinates. In this case, when you tell the mill to drill a hole one inch from the left hand side of your workpiece, you would rather use program coordinates (for example, $X = 1.000"$) than mill coordinates (for example, $X = -6.5889"$).

By moving the mill to a location on your part (often the top face, center of your part or the top left hand rear corner) and zeroing the digital readouts (DRO), you define a relationship between the mill coordinates and the program coordinates. This general term for this relationship is work offsets.

Tool Offsets

Tool offsets allow the operator to use tools of different length and, in the case of cutter radius compensation, different diameters. In the program you will create during this tutorial, you will use two different tools. Because it is extremely unlikely that these tools will be exactly the same length, the control needs to account for the difference in tool length when switching tools.

If you measure your tools when you put them in TTS holders, then the PathPilot operating system allows you to switch tools quickly and without the need to do anything more when you run a program using them. Each tool and its holder only needs to be measured once, either offline or on the mill.

Once a tool has been measured, the tool length offset must still be applied. Tool length offsets are not applied automatically – on virtually all CNC milling machines, the tool length offset is applied with the G43 command. When running a G-code program, the G43 G-code command must be called out to apply a tool length offset – tool offsets will not be applied with just a tool change command. While operating manually, the *M6 G43* button does this for you. The code you generate using the *Conversational* screens later in this tutorial will include the G43 command in the appropriate place in the G-code. Use of cutter compensation (G41/42) is a more advanced topic which is covered in chapter 7, *Programming*.

5.1.5 Set the Length Units

You can program your machine in either inches or in millimeters. The machine uses the defined setting until you program a different command (G20 or G21). The settings are also retained after a power cycle, once the machine is out of reset.

5.1.5.1 Programming in Inches

Depending on your workflow, do one of the following:

- Type `G20` in the MDI line and press *Enter* on your keyboard
- Program `G20` in your G-code program

5.1.5.2 Programming in Millimeters

Depending on your workflow, do one of the following:

- Type `G21` in the MDI line and press *Enter* on your keyboard
- Program `G21` in your G-code program

INTRO TO PATHPILOT

5.1.6 Touch Off the Workpiece to Set Work Offsets

There are many ways of conceptualizing tool and work offsets, but we use the idea of a true positive tool length to demonstrate this first part program. When using this method we will touch the face of the spindle to the top of the workpiece to set the work Z zero (see **Figure 5.4**). If you set your work Z zero using the face of the empty spindle then touch your tools off to the same work zero, the tool length offsets are equal in value to the length of the tool. True positive tool length has a few benefits over other methods (i.e., relative tool lengths) of measuring tool offsets:

- You can easily look at the tool length offset value and estimate whether it is correct for a given tool by checking that tool with a ruler or calipers.
- You can mix tools that have been touched off on the mill with tools that have been measured using a digital height gauge.
- It is conceptually easier to understand.

Setting the Z Work Offset

1. If a tool is in the spindle, remove the tool from the spindle.
2. Type 0 in the tool DRO and press the *M6 G43* button to tell the PathPilot operating system that we are changing tools and applying a tool length offset. Tool zero represents an empty spindle, and there is no offset to apply. We press the *M6 G43* button to make sure there is no tool length offset applied before we set the work offset (see **Figure 5.5**).
3. Place a piece of scrap 2" x 4" in vise. Make sure that the top of 2" x 4" is at least 1/4" above the top of the vise jaws.
4. Place a piece of paper on the 2" x 4" and jog the spindle down carefully until the spindle nose just makes contact with the top of the piece. You will be able to feel when the paper is pinched (see **Figure 5.6**).

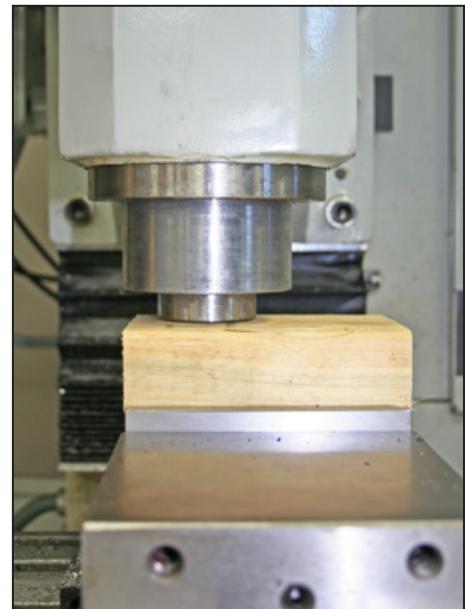


Figure 5.4

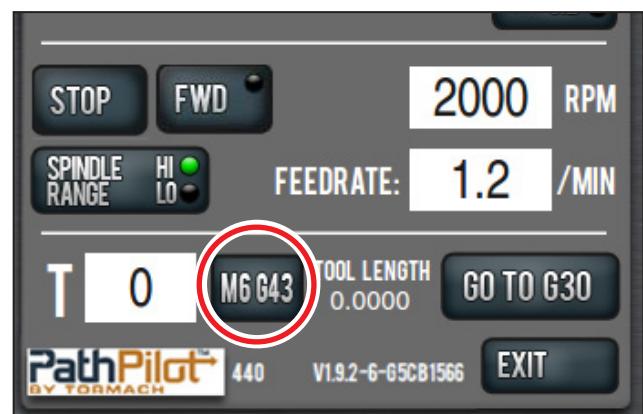


Figure 5.5



Figure 5.6

5. Click the *Zero Z* button next to the Z DRO to set the work offset Z to zero.

NOTE: This is just like typing 0.0 into the Z DRO and pressing Enter. To account for the thickness of the paper used in touching off the work offset, you could type 0.003 in the Z DRO and press Enter.

Setting the X and Y Work Offsets

Common positions for the X and Y part zeros are:

- The back left of the workpiece
- The center of the workpiece
- A feature (i.e., a hole or a boss) that already exists on the workpiece

For the first part tutorial, we will use the X/Y center of the workpiece as the zero point. To set the X and Y work offsets for this part:

1. Using a straight edge, draw two lines on the 2" x 4" from corner to corner, creating an X in the center of the workpiece.
2. Put the tool holder with the 3/8" end mill in it into the spindle.
3. Jog the mill so that the 3/8" end mill is approximately centered over the X on the workpiece.
4. Click the *Zero X* button next to the X DRO.
5. Click the *Zero Y* button next to the Y DRO.

5.1.7 Touch Off the Workpiece to Set Tool Length Offsets

This section assumes that you have already set the work offset Z zero to the top surface of the part using the steps in *Setting Work Offset by Touching off Workpiece*. The steps below describe an alternative to using the TTS height gauge. If you have the 8" Digital Height Gauge (PN 31761), it may be easier to measure the tools offline and enter their lengths directly into the tool table on the Offsets tab.

To touch off the tool offsets:

1. The 3/8" tool used to set X and Y work offsets earlier in this chapter should still be in the spindle; this is *Tool 1*. Type 1 in the tool DRO and click the *M6 G43* button to tell the mill that you have changed tools and want to apply the tool length offset.
2. Jog the mill down so that the tool just touches the top of the 2" x 4" (see **Figure 5.7**).



Figure 5.7

INTRO TO PATHPILOT

3. On the *Offsets* tab, enter *0.0* in the touch DRO and click the *Touch Z* button (see **Figure 5.8**). If you were not touching on the top of the workpiece, but instead using a feeler gauge or piece of paper between the workpiece and the tool, you could enter the thickness of the gauge or paper in the touch DRO before clicking *Touch Z* to account for the gauge thickness.
4. Look at the length value in the tool table for *Tool 1*. Verify that it is correct by measuring the length of the tool from the spindle nose to the tool tip with a ruler or calipers.
5. Enter the diameter of the tool in the tool table (see **Figure 5.9**) and press *Enter*.

NOTE: Fractions entered in these entry fields are converted to their decimal equivalents.

6. Put the $1/8"$ end mill tool holder into the spindle.
7. Type *2* in the tool DRO and click the *M6 G43* button (see **Figure 5.5**).
8. Repeat steps 3-6 to measure the tool length for *Tool 2*.

5.1.8 Write the G-code

Now use Conversational programming capabilities of the PathPilot Interface to generate G-code to produce our part (see **Figure 5.1**). This will be broken down into two operations:

1. Mill a $0.100"$ deep, $3.25"$ diameter pocket in the face of the workpiece.
2. Engrave letters *PCNC* in the pocket.

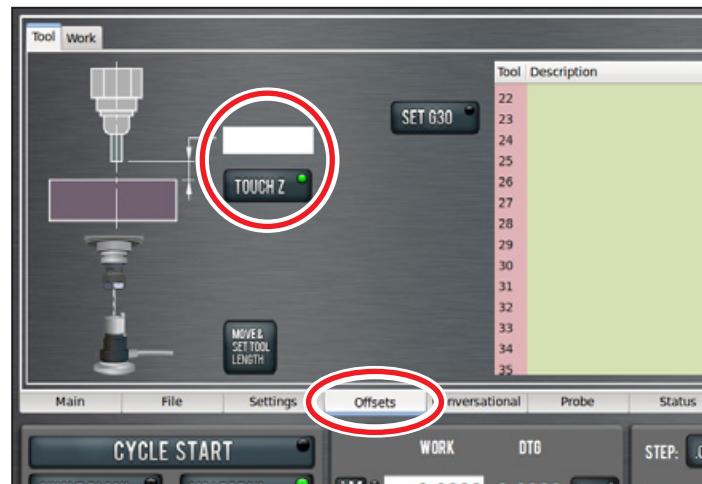


Figure 5.8

Tool	Description	Diameter	Length
1	2 flute endmill	0.5000	3.2250
2	engraver	0.1250	3.1466
3		0.0000	0.0000
4		0.0000	0.0000
5		0.0000	0.0000
6		0.0000	0.0000
7		0.0000	0.0000
8		0.0000	0.0000
9		0.0000	0.0000
10		0.0000	0.0000
11		0.0000	0.0000

Figure 5.9

5.1.8.1 Operation 1

To write the code for the first operation, click the *Conversational* tab (see **Figure 5.10**). The *Conversational* screen is divided into two sections: parameters common to most operations are displayed on the left and parameters (including part geometry) that are operation specific are displayed on the right.

Click the *Pocket* tab to bring up the pocketing screen. Click the *Rect/Circ* button to bring up the circular pocket screen (see **Figure 5.10**). Use this screen to generate code to create a shallow pocket (0.1000" deep and 3.250" in diameter). The conversational DRO fields should be self-explanatory and are covered in detail in chapter 6, *PathPilot Interface*, but for the first part tutorial, enter the values seen in **Figure 5.10**.

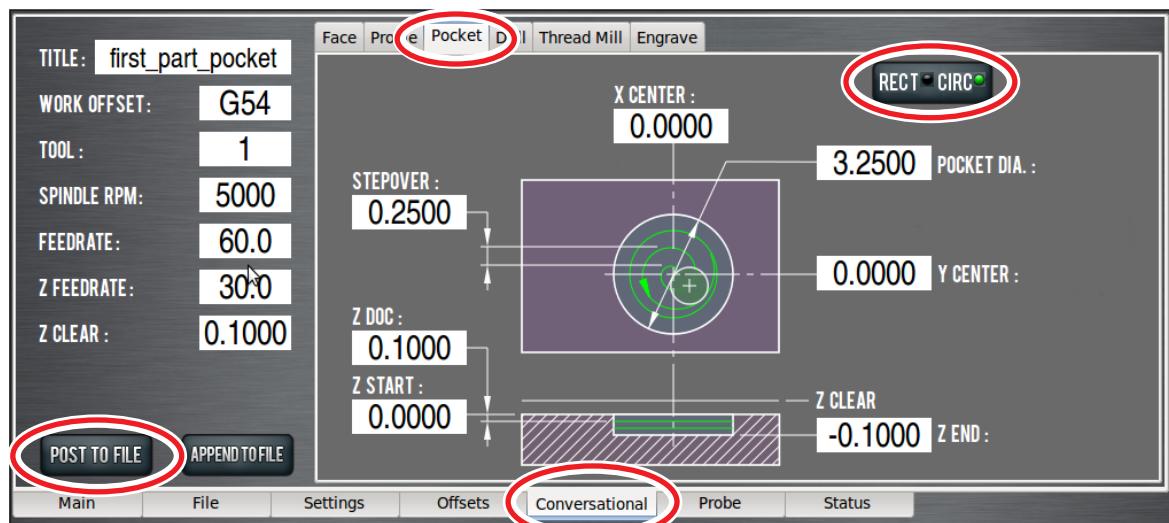


Figure 5.10

Make note of a few things:

- Units are expressed according to the current G20/21 setting. If you are in G21 (metric), the feed rates will be in mm/min and the coordinates in mm. For the purposes of this tutorial, use imperial units (G20). You can check the current G20/21 setting by inspecting the string of active G-codes next to the word *Status* at the bottom middle of the screen (see **Figure 5.12**).
- To enter values in a DRO, simply click the mouse inside the DRO and type a number, then click *Enter* on the keyboard. Pressing *Enter* in the conversational DROs is not required, but is recommended as the control will automatically move your cursor to the next DRO in the sequence and will perform validation to make sure you have not entered an illegal value.
- After the values from **Figure 5.10** are entered, click the *Post to File* button to save the G-code.
- When you click *Save* (see **Figure 5.11**), it also automatically loads into the control and displays the tool path (see **Figure 5.12**).

To run the program:

INTRO TO PATHPILOT

1. Grab the *Maxvel* slider (lower left hand corner of screen) by clicking and dragging it down to zero (see **Figure 5.12**).

NOTE: If your mill is equipped with an ATC, setting *Maxvel* to zero stops all motion and will prevent the mill from changing tools.

2. Click *Cycle Start* button (see **Figure 5.12**). If the current tool is not *Tool 1* and you have configured the mill for manual tool changes, the *Cycle Start* button LED may blink requesting a tool change. Change the tool and confirm by clicking the *Cycle Start* button again. If equipped with an ATC, tool changes happen automatically without operator interaction.



Figure 5.11

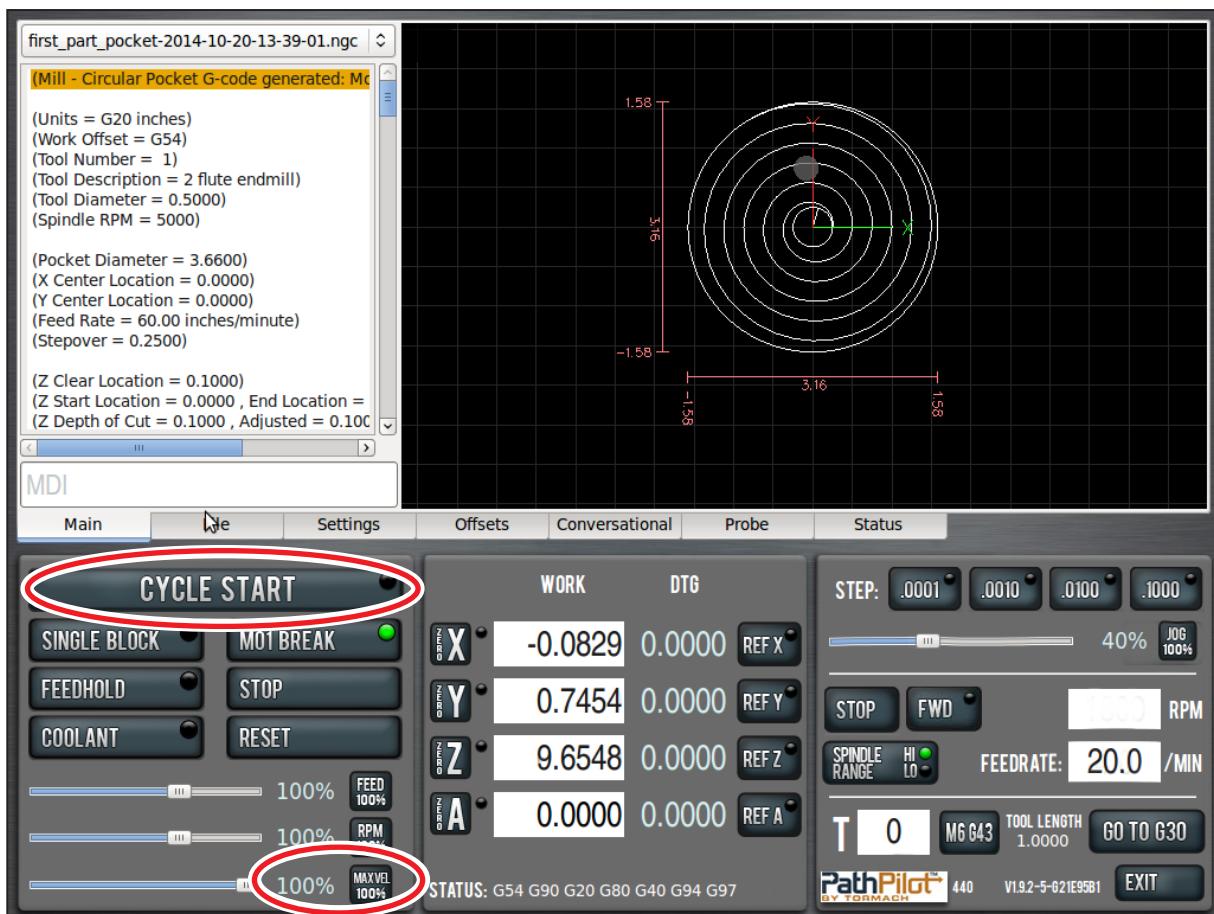


Figure 5.12

- Grab *Maxvel* slider again and slowly increase allowed velocity (see **Figure 5.12**). Bring velocity back down to zero when you get close to the part and double check values in the DROs to make sure that tool position looks correct. For example, if tool is $1/4"$ above the workpiece, Z DRO should read 0.2500. If everything looks correct, move *Maxvel* slider back up to resume part program.

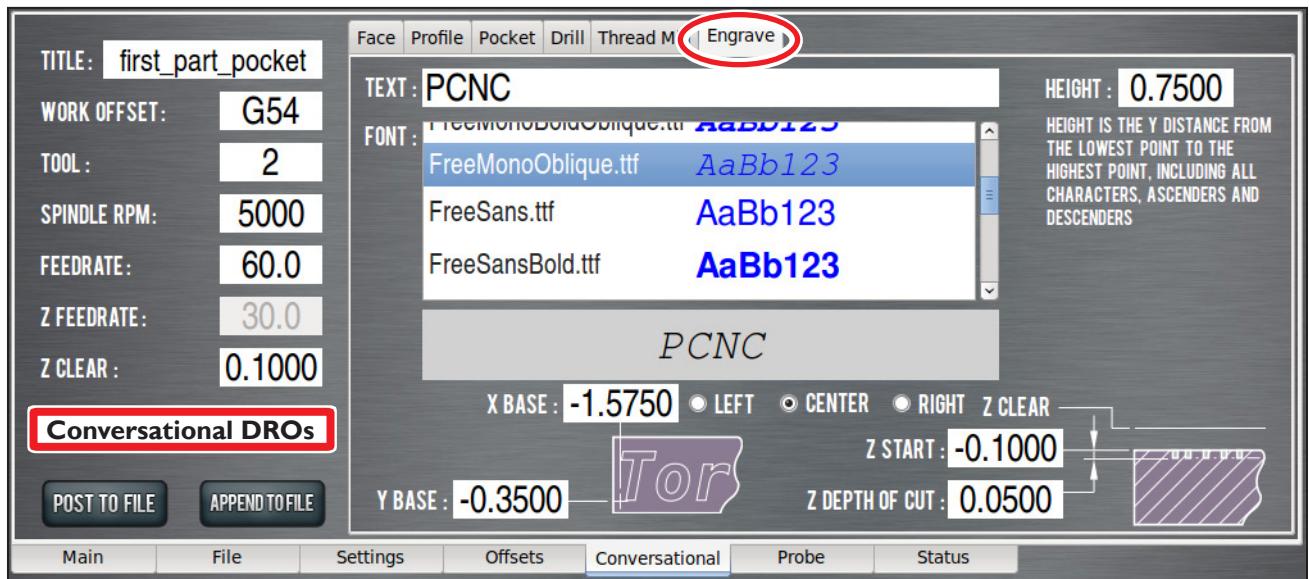


Figure 5.13

5.1.8.2 Operation 2

Go back to the Conversational pages and click on the *Engrave* tab (see **Figure 5.10**) and the Engrave screen opens (see **Figure 5.13**). Enter the values shown in **Figure 5.13**, *Conversational DROs*. Make sure to enter *PCNC* in the text field (see **Figure 5.13**).

- Select the *FreeMonoOblique.ttf* font (see **Figure 5.13**) and change the tool DRO to 2 for the engraving operation.
- Click the *Append to File* button (see **Figure 5.13**). A file chooser dialog opens that allows you to select the file to which you want to add the engraving G-code.

INTRO TO PATHPILOT



Figure 5.14

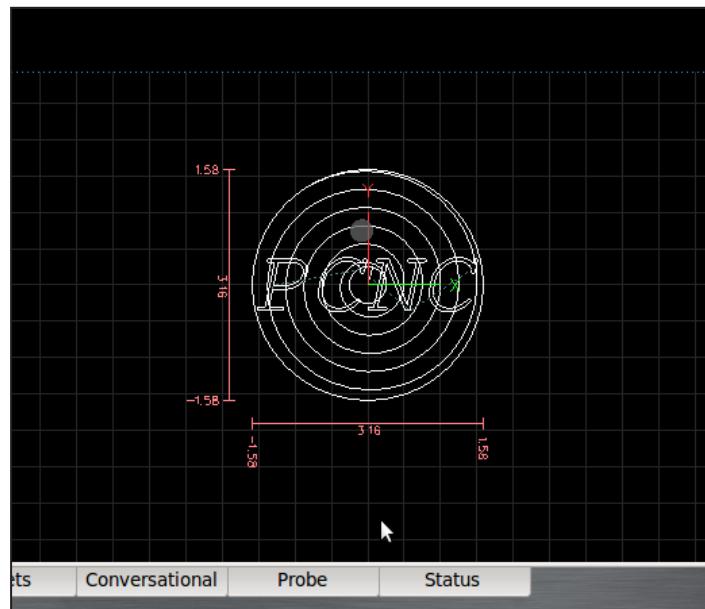


Figure 5.15

3. Click on the name of the file you created when you made the facing G-code, then click *Append to File* (see **Figure 5.14**). The changes to your file are loaded into the control, and you should see a tool path that looks something like **Figure 5.15**.
4. Run the completed program using the method described in section *Operation 1* earlier in this chapter.

NOTE: When you run this code, it will recut the pocket that you created in the first operation. If you wanted to, you could have posted this code to a separate file.

PATHPILOT INTERFACE

6. PATHPILOT INTERFACE

6.1 Overall Layout

The PathPilot® interface is divided into two sections: *Notebook* and *Persistent Controls* (see Figure 6.1). *Persistent Controls* make up the bottom half of the screen with three control groups: *Program Control Group*, *Position Status Group*, and *Manual Control Group*. The top half of the screen is the *Notebook*, which includes seven tabs: *Main*, *File*, *Settings*, *Offsets*, *Conversational*, *Probe*, and *Status*. Depending on the mill accessories, there may also be optional tabs including *ATC* (automatic tool changer), *Injection Molder*, and *Scanner*. These tabs are used to select different *Notebook* pages, each of which displays various buttons, digital readouts (DROs), and information pertinent to the functioning of the PathPilot interface.

PathPilot Interface

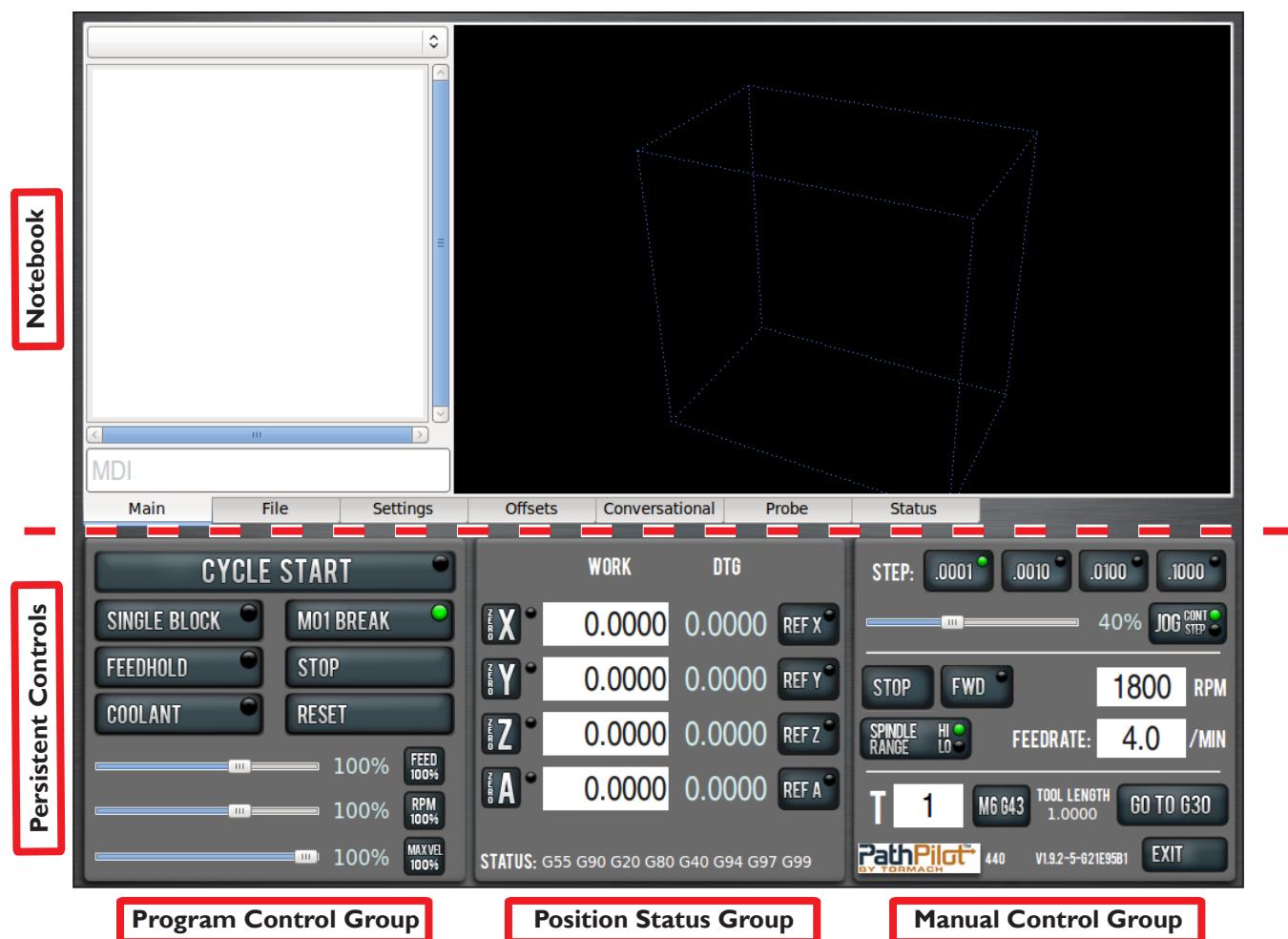


Figure 6.1

PATHPILOT INTERFACE

For example, the *File* page of the Notebook is used for tasks like transferring a G-code file from a USB drive to the controller, loading a G-code file into memory, or editing a G-code file.

While the *Notebook* half of the screen allows you to perform a variety of tasks based on which tab is active (loading G-code file, writing G-code with the *Conversational* tab, touching off tools), the *Persistent Controls* half of the PathPilot interface contains the controls used to set up a job and execute G-code. Operators already familiar with Tormach milling machines (or most other CNC machines) will be familiar with many of the *Persistent Controls* buttons.

For definitions and detailed information on the terminology used, refer to chapter 7, *Programming*.

6.2 Persistent Controls

The *Persistent Controls* on the lower half of the screen are always present – they don't move or disappear as you page through the *Notebook* (see **Figure 6.1**). These are divided into three logical families: *Program Control Group*, *Position Status Group*, and *Manual Control Group*.

6.2.1 Program Control Group

The buttons, sliders, and DROs of the *Program Control Group* are functions that relate to tasks performed while running a G-code program (see **Figure 6.2**). They may be used at any time while running a program, or before running a program to set modes like *Single Block* or *M01 Break*.

Cycle Start – Starts a program. While running a program, the LED in the upper right hand corner of the button illuminates.



Figure 6.2

If *Single Block* is active (for more information, refer to *Single Block* later in this section), *Cycle Start* causes the mill to execute one line of G-code per click. When running a program, if motion is paused due to *Feedhold*, *M01 Break*, *Single Block*, or because the mill is waiting on a manual tool change, the *Cycle Start* button LED flashes until the *Cycle Start* button is pressed again.

It is an error if:

- *Cycle Start* is clicked when the *Main* tab of the notebook is not active
- *Cycle Start* is clicked when no G-code program is loaded
- *Cycle Start* is clicked before the mill has been referenced

Single Block – Turns *Single Block* on (LED illuminated) or off. When *Single Block* mode is active, the mill executes one block of G-code, then pauses and flashes the *Cycle Start* button LED, indicating that *Cycle Start* needs to be clicked to execute the next line of G-code. This feature may be turned on or off before running a program or during program execution.

NOTE: Non-motion lines are ignored by *Single Block* mode. This means that the PathPilot interface will skip comment lines and blank lines.

M01 Break – Turns *M01 Break* on (LED illuminated) or off. When *M01 Break* is active, and an M01 (optional stop) is programmed in the G-code file, the mill stops when it reaches the M01 line and the *Cycle Start* button LED flashes. The mill continues to execute the program lines after the M01 when *Cycle Start* is clicked. This feature may be turned on or off before running a program or during program execution.

Feedhold – Turns *Feedhold* on (LED illuminated) or off. Turning *Feedhold* on pauses mill motion, and the *Cycle Start* button LED flashes. Turning *Feedhold* on leaves the spindle running (if it is already on). To turn *Feedhold* off, click *Cycle Start*. The *Feedhold* button works during program execution or during manual data input (MDI) moves (for more information, refer to *Main Tab* section later in this chapter). *Feedhold* has no effect when the mill is not moving. The feedhold function is also connected to the keyboard's space bar – pressing the spacebar on the keyboard is equivalent to clicking this button with the mouse.

Stop – Stops all mill motion, including spindle motion. If clicked while running a program or during an MDI move, the *Stop* button stops the mill and rewinds the G-code program. *Stop* doesn't change the current modal state of the mill (G54, G01, etc.).

Coolant – Turns coolant on (LED illuminated) or off. Clicking this button turns power on or off to the coolant relay in the electrical cabinet. This button is the equivalent to M8/M9 G-code commands. It may be clicked before, after, or during program execution, or an MDI move.

Reset – Brings the mill out of an E-stop condition, resets G-code modalities, clears alarm messages, and rewinds the G-code program. When the mill is first powered on, or after an E-stop, the *Reset* button flashes. When this button is flashing (after power has been restored to the mill), clicking *Reset* starts and verifies communication between the mill and the controller. *Reset* may be clicked any time after the mill is powered on. *Reset* does the following:

- Resets all modal G-codes to their normal state including work offset to G54 default
- Rewinds a G-code program
- Stops a program, MDI move, or homing move if one is currently in progress
- Clears alarms (for more information on alarms, see the *Status* tab section)
- Clears the tool path backplot

PATHPILOT INTERFACE

Spindle Override – The *Spindle Override Slider* and *RPM 100%* button allow you to override the commanded spindle speed by percentages ranging from 1 percent to 150 percent. The *RPM 100%* button returns the override to 100 percent of the commanded value or no override. The spindle must be running for these controls to have a noticeable effect. If you override the spindle when it is stopped, the speed is overridden the next time spindle starts. The override doesn't drive the spindle past its maximum speed. The *Spindle Override* setting is ignored during spindle-synchronized moves or any time M48 (disable feed and speed overrides) is in effect.

Feedrate Override – The *Feedrate Override Slider* and *Feed 100%* button work similarly to the *Spindle Override* controls. They affect the commanded feedrate by a percentage ranging from 1 percent to 150 percent. The feedrate override works for MDI, jogging, and G-code program G01/G02/G03 moves. The override has no effect on G00 (rapid) moves. The *Feedrate Override* setting is ignored during spindle-synchronized moves or any time M48 (disable feed and speed overrides) is in effect.

Maxvel Override – The *Maxvel Override* and *Maxvel 100%* button work similarly to the *Feedrate Override* controls, except that these controls affect both G00 and G01 moves. They clamp the mill velocity to a percentage of the maximum velocity. The *Maxvel Override* slider can be very useful when running a G-code program for the first time. You can use it to stop the mill by sliding it down to 0% and verifying the Distance to Go (DTG) and X/Y/Z/A DROs look appropriate before continuing (for more information, refer to *Position Status Group* later in this chapter). The *Maxvel Override* is a safety feature, and as such is not inhibited during spindle-synchronized moves or with M48. Make sure that Maxvel is at a value that allows the mill to achieve the programmed feed rate during spindle-synchronized moves or the move may fail to produce the intended results.

6.2.2 Position Status Group

The buttons, labels, and DROs of the *Position Status Group* pertain to mill position, active G-code modalities, and feed/speed settings (see **Figure 6.3**). These controls may be used at any time before or after running a G-code program or MDI move. They are unavailable for operator input while mill is moving.

Axes Work Offset DRO – Displays the current mill position expressed in the currently active work offset coordinate system (G54, G55, etc.) for the X-, Y-, Z-, and A-axis.

When the mill is at rest, these DROs are also operator entry fields. Change the current work offset position by clicking in the DRO field, which illuminates. Type a number, for example 4.0, and click the *Enter* key. Click the *Esc* key to return to the original value.

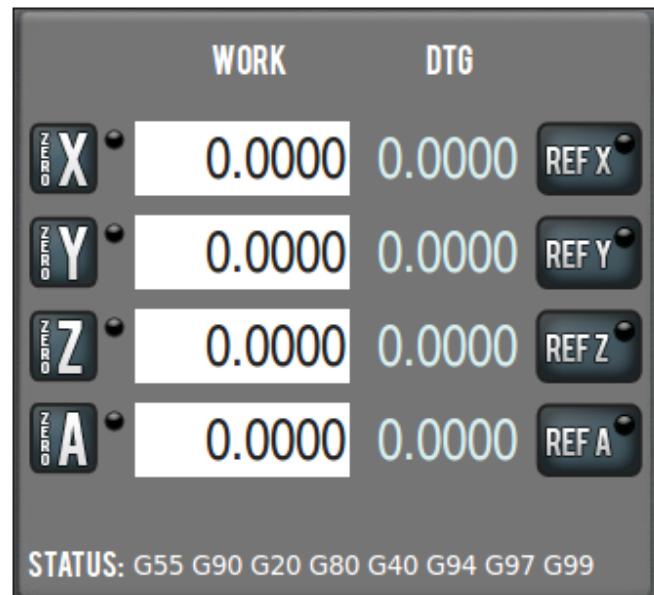


Figure 6.3

PATHPilot INTERFACE

This technique is used for setting any DRO. Remember to click *Enter* after any DRO change. If you forget and just click on another DRO field, any value you have just entered is discarded. This is designed to avoid accidental changes.

For convenience, the *Zero* button to the left of the axis DROs can be used to set the current work offset position for that axis to 0.000.

Distance to Go (DTG) – Just to the right of the axis DROs are the *Distance to Go (DTG)* labels in light blue. These labels are read-only and display the distance remaining in any single move. If you feedhold the mill in the middle of a move, or turn the *Maxvel* or *Feedrate* overrides to 0 percent, these labels display the distance left in the commanded move. These labels are useful when proving out a part.

Ref Axes Buttons – *Ref X*, *Ref Y*, *Ref Z*, and *Ref A* move the axes to their respective reference switch locations. This must be done after power on and before running a part program or using MDI commands. The axes may be referenced simultaneously, though it is common practice to reference the Z-axis first to clear the spindle or tool from the area of the workpiece or vise. When referenced, the LED illuminates.

Status – The *Status* line displays the currently active G-code modalities and the active tool. A more detailed description of these active G-codes is provided on the *Settings* tab.

Jog Active LEDs – Between the *Zero* button and DROs are LEDs. If the mill is equipped with an optional Jog Shuttle (PN 30616), the active jog axis is indicated by an illuminated LED.

6.2.3 Manual Control Group

The *Manual Control Group*'s buttons, slider, and DROs allow the operator to perform tasks related to manual control of the mill, including jogging the mill axes, changing the current tool number, feed rate, or spindle speed, and starting or stopping the spindle (see **Figure 6.4**).



Figure 6.4

PATHPILOT INTERFACE

Jog Shuttle

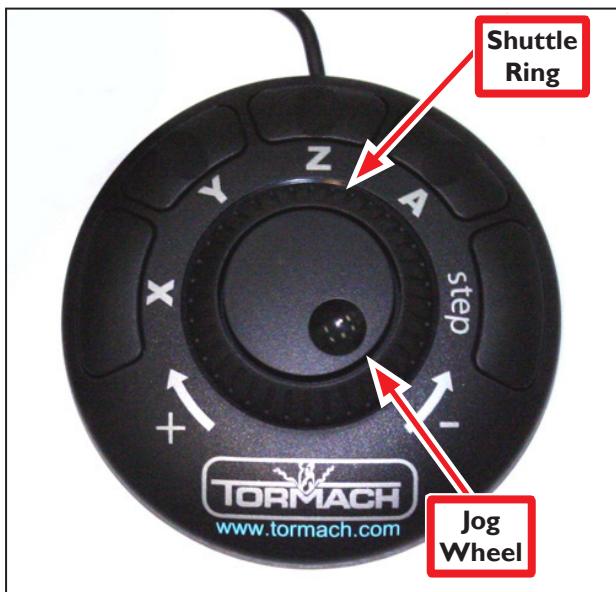


Figure 6.5

Jogging with Keyboard Keys

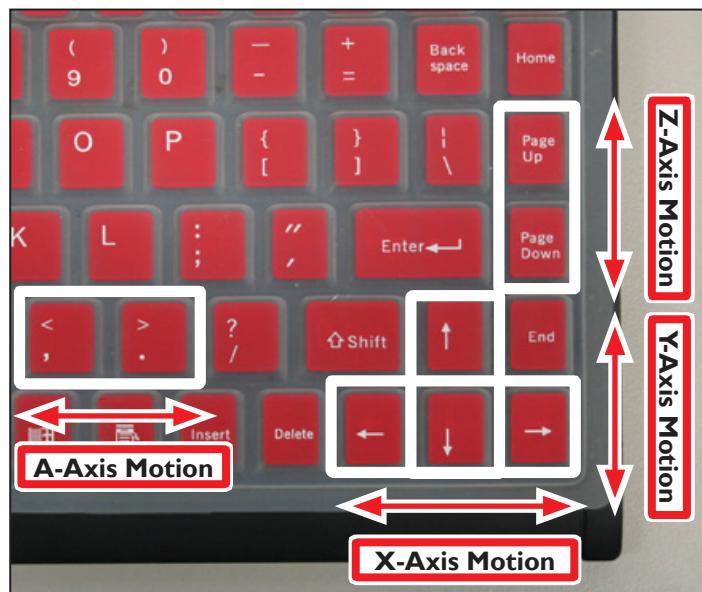


Figure 6.6

Jogging Controls – The mill can be jogged with either the optional Jog Shuttle shown in **Figure 6.5** or with the keyboard (see **Figure 6.6**):

- The *right arrow* key jogs X-axis in the positive X direction (moves table left of operator)
- The *left arrow* key jogs X-axis in the negative X direction (moves table right of operator)
- The *up arrow* key jogs Y-axis in the positive Y direction (moves table towards operator)
- The *down arrow* key jogs Y-axis in the negative Y direction (moves table away from operator)
- The *Page Up* key jogs the Z-axis in the positive Z direction (moves spindle up)
- The *Page Down* key moves the Z-axis in the negative Z direction (moves spindle down)

NOTE: *Jogging is not permitted during G-code program execution or during MDI moves.*

The Jog Shuttle is an optional accessory (see **Figure 6.5**) that may increase productivity, especially on short-run jobs requiring extensive setting up of the workpiece and tooling.

The *X*, *Y*, *Z* and *A* buttons are used to jog the *X*-, *Y*-, *Z*-, and *A*-axis, respectively. The LED light beside an axis DRO indicates which axis is selected for jogging (for more information, refer to *Position Status Group* section earlier in this chapter). The *Step* button cycles through the available jog step sizes. The size that is active is indicated by an illuminated LED (see **Figure 6.4**). Continuous jogging is done with the Shuttle Ring by turning it counterclockwise (minus) and clockwise (plus). There are seven speeds to position any axis with speed and precision. Step jogging is done with the Jog Wheel (with finger dimple) by turning it counterclockwise in the minus direction and clockwise in the plus direction. The move will be made at the current feed rate.

PATHPILOT INTERFACE

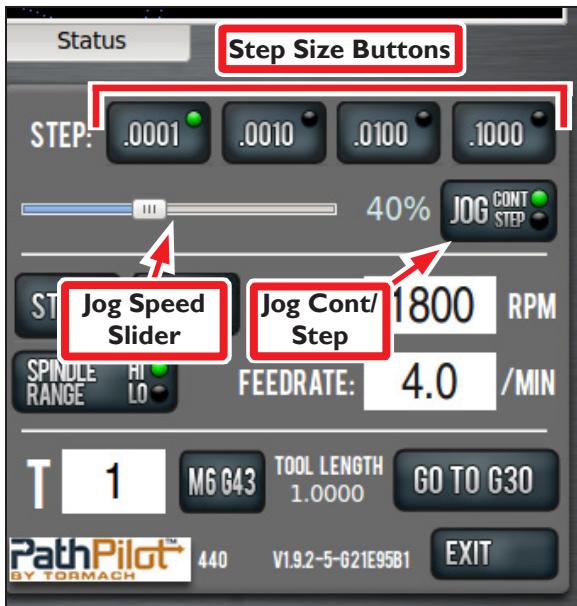


Figure 6.7



Figure 6.8

Step and Continuous Jogging – Whether using the jog shuttle or the keyboard, there are two modes of jogging: continuous and step. When using the keyboard to jog, switch between modes using the *Jog Cont/Step* button (see **Figure 6.7**).

In *Step* mode, the mill jogs in steps, where the step size is controlled by the four buttons to the right of the *Step* label (see **Figure 6.7**).

In *Continuous* mode, the mill jogs at a continuous velocity when you press and hold any available keyboard key; stop the mill by releasing the key. Axis motion is key-specific as shown in **Figure 6.6**. The velocity is set using the *Jog Speed Slider* (see **Figure 6.7**). To set jogging velocity to the maximum speed, click and drag the *Jog Speed Slider* to the far right position.

Notice that in imperial units (type G20 in the MDI line), the step sizes range from 0.0001" to 0.1" (see **Figure 6.7**), whereas in metric mode (type G21 in the MDI line) the step sizes range from .01 mm to 10 mm. The illuminated LED in the upper right corner of each *Step Size Button* indicates active step size (see **Figure 6.7**).

Feedrate DRO – Feedrate is the velocity at which the workpiece can be fed against the machine tool. The DRO is the field that displays this velocity.

Spindle Controls – *Stop* and *FWD* can be used to manually control the spindle (see **Figure 6.8**). The *Stop* button stops the spindle, similar to the M5 command. The *FWD* button starts the spindle clockwise at the set RPM.

These buttons are unavailable when running a G-code program or in the middle of an MDI move. Clicking *FWD* triggers an alarm if the commanded spindle speed is outside of the valid spindle speed range for the mill's current belt position.

PATHPILOT INTERFACE

The Spindle RPM DRO is used to display the current spindle speed command (see **Figure 6.8**). Change the current spindle speed command by typing a value into the DRO and pressing *Enter*. Values above the maximum RPM or below the minimum RPM for the current belt position triggers an alarm.

The *Spindle Range* button toggles between the two belt/pulley settings with an LED indicating which position is active (see **Figure 6.8**). For more information on spindle belt/pulley settings, refer to chapter 4, *Operation*.

Tool DRO (T) – Displays the tool currently in the spindle. To change the tool in the spindle and apply its tool length offset, type a number (valid range is 0-256) into the tool DRO and click either the *Enter* key or the *M6 G43* button.

M6 G43 Button – Causes the system to change the number of the tool that is currently in the spindle to the number typed in the DRO, as well as apply the length offset for that tool. M6 is the G-code command that requests a tool change, and G43 is the command that applies a tool length offset (for more information on these commands, refer to chapter 7, *Programming*).

Tool Length Label – Displays the current tool length offset. This display is normal (light blue text on grey background) when the tool offset number matches the tool number; an alarm appears (orange text on red background) if the offset number does not match the current tool number.

Go to G30 Button – Causes the mill to move to the a pre-defined G30 position, and is equivalent to typing G30 in the MDI line. This G30 position can be set using the *Set G30* button on the *Offsets* tab. Operators familiar with M998 will notice that the behavior of G30 is identical to M998. By default, the move to the G30 position is in Z only. This can be changed on the *Settings* tab.

To set the G30 position, jog the mill to the desired position and click the *Set G30 Position* button on the *Offsets* tab. Subsequent uses of the G30 command in G-code or the *Go To G30* button will cause the mill to move to this position.

6.3 Keyboard Shortcuts

Several keyboard shortcuts are provided for operator convenience. Below is a list of shortcuts used in the PathPilot interface:

Spacebar	Feedhold	Alt+R	Cycle Start
ESC	Stop	Alt+F	Coolant
Alt + Enter	Give focus to MDI line	Alt+E	Edit currently loaded G-code program ¹

¹Use the Alt+E command on any PathPilot screen to edit G-code.

6.4 Main Tab

The *Main* tab is active by default when the PathPilot controller first powers on, and contains four controls: recent files, G-code window, MDI line, and tool path display (see **Figure 6.9**).

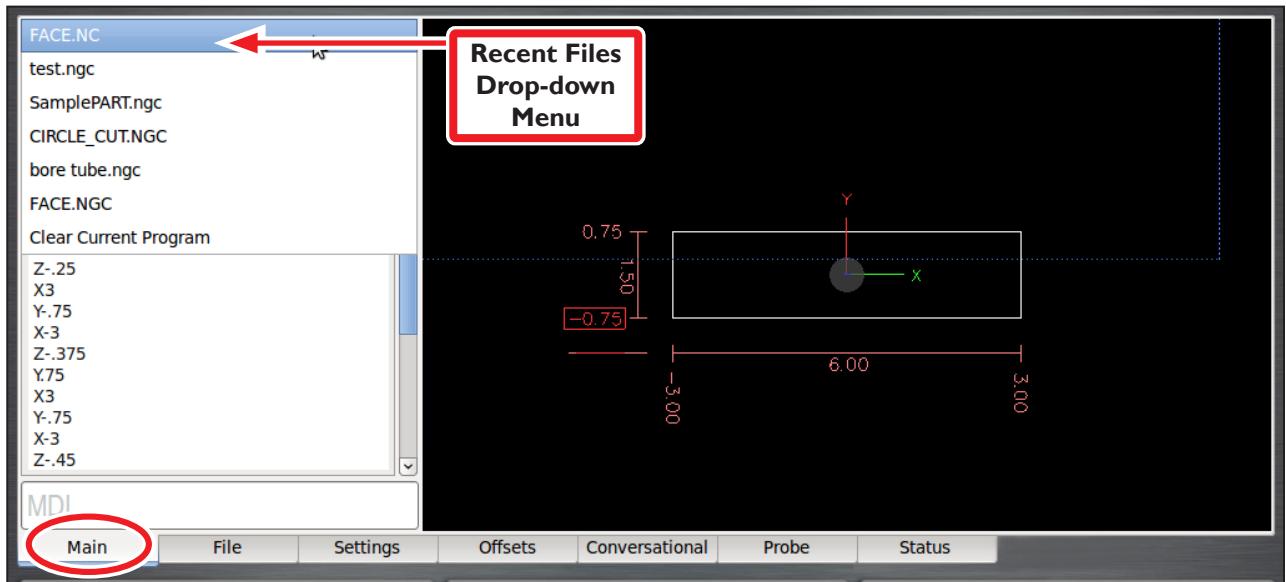


Figure 6.9

6.4.1 Selecting a Recent G-code Program File

The recent files drop-down menu displays the currently loaded G-code program file (see **Figure 6.9**). Click the drop-down menu to display the last five program files loaded into PathPilot; select the name of the program from the menu to load the G-code.

Select *Clear Current Program* in the recent files drop-down menu to close the currently loaded G-code program file.

6.4.2 Working in the G-code Window

The G-code window displays the G-code of the currently loaded program file. Use the scroll bars to view the entire file.

PathPilot highlights certain lines of code of interest. When running a G-code program in single block mode, there may be as many as three lines of G-code highlighted, each with a different color:

- Green line: indicates the start line, which is the first line in the program unless this has been changed with the *Set Start Line* feature
- Blue line: indicates the line of code that is currently executing
- Brown line: indicates the move that will occur the next time *Cycle Start* is pressed

PATHPILOT INTERFACE

6.4.2.1 Setting a New Start Line

The start line is the line at which the G-code program begins. By default, this is the first line of code in the G-code program. Right-click the preferred start line of code in the program and select *Set Start Line* to change the start line.

When using the *Set Start Line* option, the operator is responsible for ensuring that the mill is in the proper state before the code executes.

To set a start line in the middle of a G-code program file, make sure any preparatory moves (like turning the spindle and coolant on) are manually completed before clicking *Cycle Start*.

The mill reads backwards through the beginning of the G-code program file to do things like set the appropriate G5x active work offset, G61/64 setting, and other modal states. It will not turn the spindle or coolant on.

6.4.2.2 Expanding the G-code Window

Double-click the G-code window to expand the G-code window and shrink the tool path display. Double-click the G-code window again to return the display to its original size.

6.4.3 Manually Entering Commands

When running a G-code program, commands to the mill are read from a file. You can also send G-code commands to the mill directly with the MDI line (see **Figure 6.10**).

Click the *MDI* field to use the MDI line; the line is highlighted. Type the command, using the *Backspace*, *Delete*, *Left* and *Right* arrow keys to correct typing errors. Press *Enter* to execute the command; press *Esc* to abandon it.

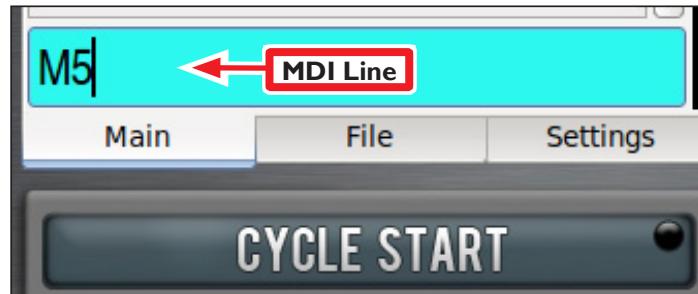


Figure 6.10

Click the *MDI* field and use the *Up* or *Down* arrow keys to copy a recent MDI command into the MDI line. Up to 100 MDI commands are stored for reuse; these commands are saved between sessions. Command history is available after a power cycle.

NOTE: When the MDI line is open, all keystrokes are registered as a typed command. Jogging is not possible when you are clicked inside the MDI field.

6.4.3.1 Searching in the Code

MDI has the ability to search the text of a G-code program file for specific numbers, codes, or items of interest like tools, feeds, and speeds.

PATHPilot INTERFACE

Type *FIND* followed by the text to be searched in the MDI line (see **Figure 6.11**). Pressing *Enter* finds the next instance of the searched text; pressing *Enter* while holding down the *Shift* key finds the previous instance.

If found, PathPilot scrolls to the line containing the searched text and highlights it in yellow (see **Figure 6.11**). When the search reaches the end of the G-code file, it wraps and starts again from the beginning.

Change the starting point of the search by clicking on any line in the G-code window.

When used in conjunction with the *FIND* command, certain search terms (listed below) initiate a search through the G-code file to find more than just the actual search term:

- *FIND TOOL*: Searches for instances of the actual word *Tool* in the G-code and any *T* G-code command which calls up a tool (e.g., T12)
- *FIND SPEED*: Searches for instances of the actual word *Speed* in the G-code and any *S* G-code command
- *FIND FEED*: Searches for instances of the actual word *Feed* in the G-code and any *F* G-code command (see **Figure 6.12**)

NOTE: Search text ignores case, so the command *FIND TOOL* will match *TOOL*, *Tool*, *tool*, etc.

The *FIND* command simplifies searching of a G-code file to verify speed and feed values and tool calls before cutting a part, or to find a specific set start line point in a large G-code file. For more information on using set start line, refer to *Setting a New Start Line* section earlier in this chapter.

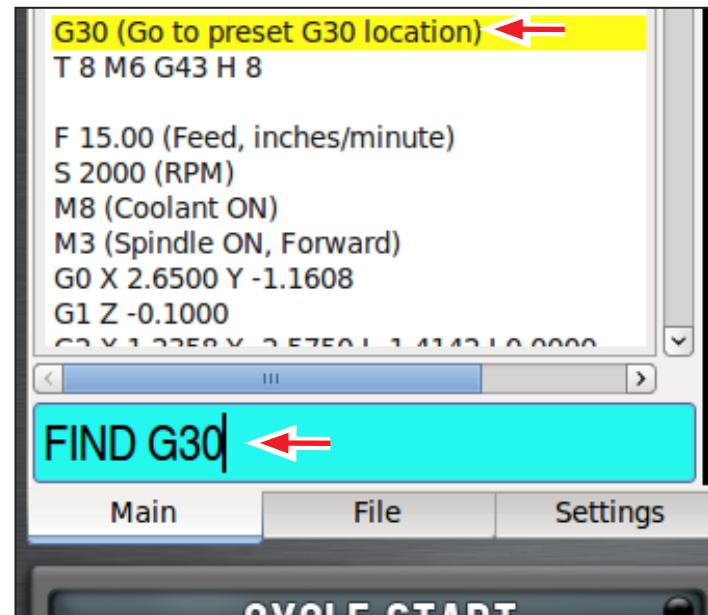


Figure 6.11

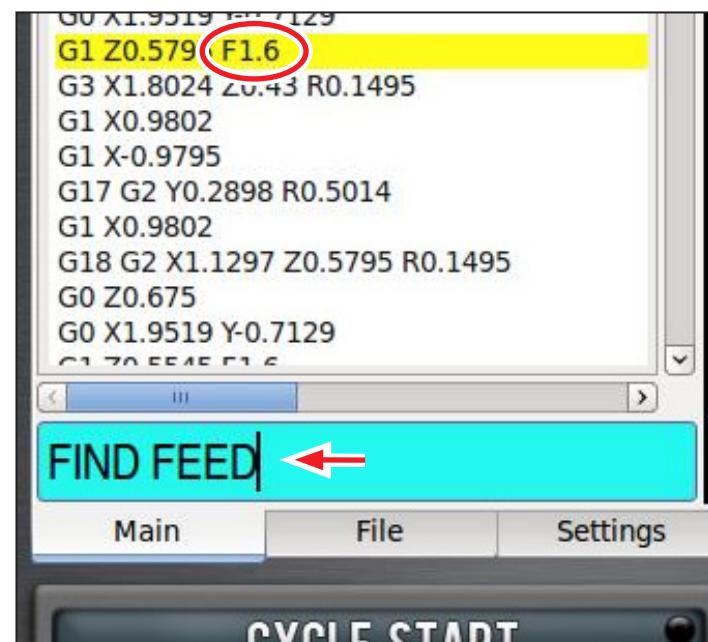


Figure 6.12

PATHPILOT INTERFACE

6.4.4 Working in the Tool Path Window

The tool path window displays a graphic representation of the tool path that is executed for the currently loaded G-code file (see **Figure 6.13**), each with a different color:

- White lines: indicates the preview lines
- Red lines: indicates the tool path as it is cut
- Yellow lines: indicates jogging moves
- Dotted blue lines: indicates the boundary box, which represents the ends of travel of the axes

Double-click the tool path window, or click *Reset*, to erase the jogging or tool path lines.

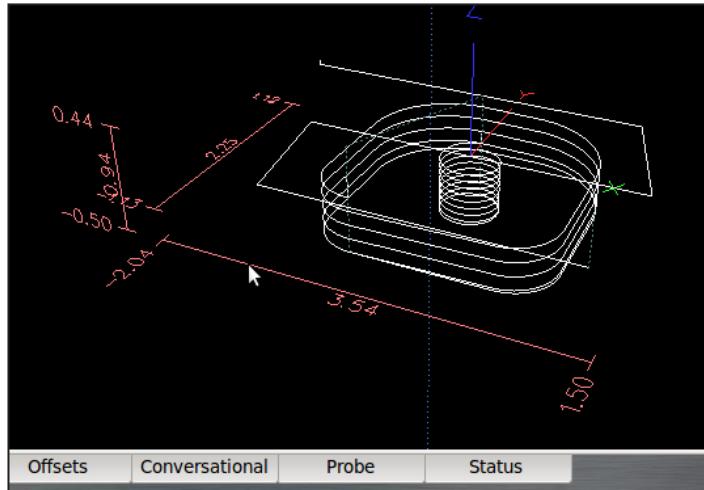


Figure 6.13

6.4.4.1 Changing the View of the Tool Path Window

Four views are available: top, front, right, and ortho. By default, the view is top. Right-click anywhere in the tool path display and select a different view to change the view of the window.

Grid lines are visible behind the tool path when the view is top, front, or right. Grid lines are not available in ortho. By default, grid lines are drawn at 0.5" intervals when in G20 mode (5 mm intervals when in G21 mode). Right-click anywhere in the tool path display and select a different grid spacing to change the resolution of the grid lines. When a program is loaded, the program extents (furthest points to which the tool will travel while executing the G-code) are displayed to the left and bottom of the tool path (see **Figure 6.13**).

6.5 File Tab

The *File* tab is used to transfer files to and from a USB drive, copy, delete, and rename files and folders (see **Figure 6.14**). The left window shows files and folders on the controller hard drive; the middle window shows files and folders on a removable USB drive.

NOTE: The controller does not run programs from the USB drive – programs must be copied to the controller before loaded and run.

6.5.1 Managing Files

Use the *New Folder*, *Rename*, and *Delete* buttons below the respective USB Drive Window and Hard Drive Window for file management (see **Figure 6.14**). To move files into a folder, right-click on the file and select cut or copy from the pop-up menu.

PATHPilot INTERFACE



Figure 6.14

6.5.1.1 Transferring Files or Folders from a USB Drive

1. Insert a USB drive into any open USB port.
2. Navigate to the file to transfer USB drive window.

NOTE: Use Back to navigate backwards; use USB to jump to the highest (home) level (see Figure 6.14).

3. In the hard drive window, navigate to the desired location in the PathPilot interface to copy the transferred file from the USB drive.
4. Highlight the file or folder to copy in the USB drive window; click *Copy From USB* (see Figure 6.14).
5. If the file to be transferred has the same name as an existing file on the controller, you can either overwrite the file, give it a different name, or cancel the file transfer.
6. When copied to the new location, the file displays in the USB drive window.
7. Click *Eject* to disconnect the USB drive from the controller (see Figure 6.14).

NOTE: Ejecting the USB drive this way helps to avoid corrupting data on the USB drive.

6.5.2 Loading G-code

The *Load G-code* function is only available for files stored on the controller (see Figure 6.14).

1. Navigate to the desired .nc file in the hard drive window; highlight the file and click *Load G-code* (see Figure 6.14).
2. Click on the *Main* tab; verify G-code file name appears in recent files drop-down menu. For more information on selecting a recent G-code program file, refer to *Main Tab* section earlier in this chapter.

PATHPILOT INTERFACE

6.5.3 Editing G-code

The *G-code File Preview* window displays the contents of the selected .nc file (see **Figure 6.14**). You can edit G-code in two ways on your PathPilot controller:

- Using a text editor
- Using the *Conversational* tab to edit a file created in PathPilot's conversational programming

NOTE: For more information on using conversational programming in PathPilot, refer to *Conversational Tab* section later in this chapter.

6.5.3.1 Editing G-code with a Text Editor

1. Highlight the file and click *Edit G-code* (see **Figure 6.14**).
2. A text editor opens the file in a new window for editing the contents of the file. Make the appropriate changes to the file and click *Save*.
3. Click the *X* in the upper right-hand corner of the screen to close the text editor.
4. Click *OK* when asked to re-load the file.

6.5.3.2 Editing G-code with Conversational Programming

To edit the G-code using conversational programming, highlight the file and click *Conv. Edit* (see **Figure 6.14**). A job assignment editor opens the file in a new window for editing the contents of the file. The left window shows job assignments of a program; the right window previews the program (see **Figure 6.15**).

Use *Move Up*, *Move Down*, *Duplicate*, or *Remove* to edit the order in which the steps of the program occur.

Click *Insert Step* to create a new job assignment using conversational programming. PathPilot opens the *Conversational* tab; after creating the job assignment, click *Insert*. If necessary, edit the order of the newly created job assignment in the program.

Click *Insert File* to load an already created G-code file into the program – allowable G-code files can be hand-written, generated from a CAM software, or generated from PathPilot's conversational programming. Navigate to the desired .nc file and highlight; click *Open*. If necessary, edit the order of the newly created job assignment in the program.

Highlight a job assignment and click *Conv. Edit* to edit a job assignment created in conversational programming. PathPilot opens the relevant tab of the *Conversational* tab; make the appropriate changes to the job and click *Finish Editing* to update the G-code program file.

Click *Revert* to restore any edited job assignment to its original parameters. *Revert* is only available for individual job assignments created in conversational programming.

Click *Close* and, when prompted, *Close Without Saving* to restore the entire G-code program file to its original parameters. PathPilot exits the conversational edit screen without overriding the original G-code program file.

PATHPILOT INTERFACE

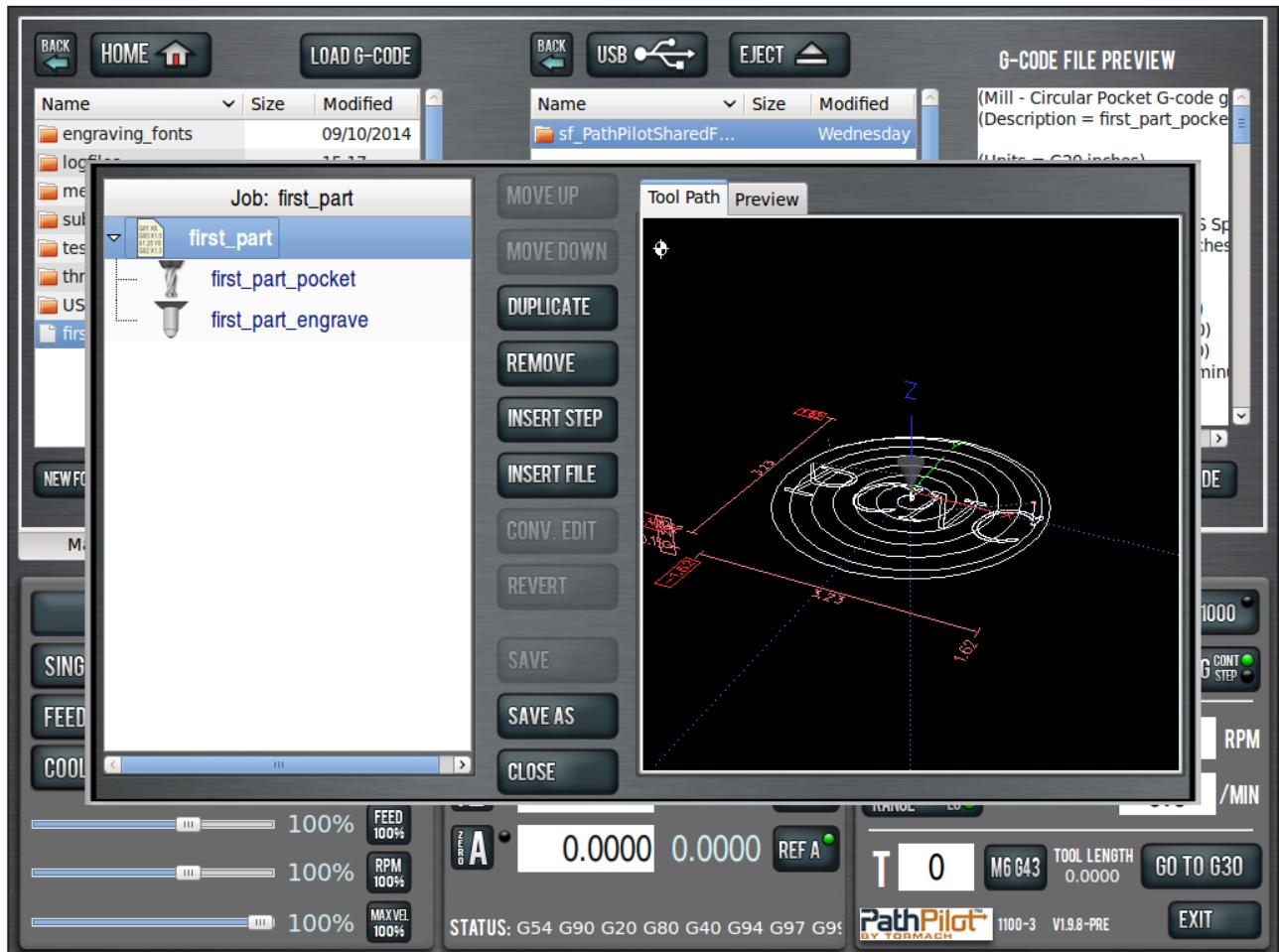


Figure 6.15

6.6 Settings Tab

The *Settings* tab displays active settings of the PathPilot controller, allowing you to configure PathPilot to suit your machine configuration (see **Figure 6.16**).

The window on the left side of the *Settings* tab displays a list of available G-code modalities. Active G-codes are highlighted in yellow (see **Figure 6.16**).

6.6.1 Selecting the Tool Changer Type

Select the tool changer type that matches mill configuration. The mill behavior when it encounters an M6 command is different depending on which tool changing option is selected on this screen.

PATHPILOT INTERFACE

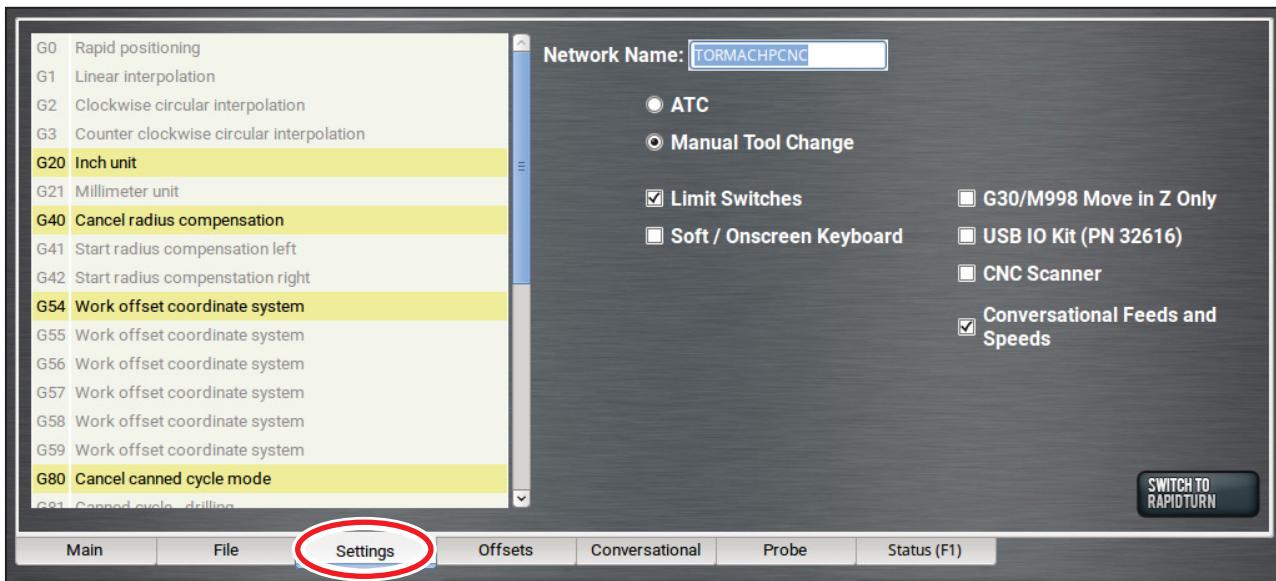


Figure 6.16

If the *Manual Tool Change* option is selected, the mill pauses at the M6 command during a G-code program file execution, allowing you to manually change tools (see chapter 4, *Operation*, for information on the manual tool change procedure). After changing tools, press *Cycle Start* to resume program execution with the new tool. When the mill is paused and waiting for a manual tool change in the middle of a G-code program, the *Cycle Start* LED flashes on and off and a message is displayed with the requested tool number on the tool path display.

6.6.2 Changing the Network Name

Use the *Network Name* field to change the network name of the controller; press *Enter*. The controller must be restarted for the name change to take effect.

If you are connected to a network using either the Ethernet jack or the optional Wireless Network Adapter (PN 34705), the controller appears on a network as *network-attached storage*. The controller exports a Windows network share named *gcode* to the Windows network with a domain name *WORKGROUP*. The default network name of the controller is *TORMACHPCNC*. The login name for the share is *WORKGROUP\operator* and the password is *pcnc*. The network name must be unique on your network. For more information, refer to the documentation that ships with the Wireless Network Adapter.

6.6.3 Disabling Reference Switches

The *Limit Switches* checkbox is selected by default. To provide a temporary workaround for a malfunctioning limit switch circuit, clear the checkbox. When cleared, *Ref X*, *Ref Y*, *Ref Z*, and *Ref A* sets the machine reference position to the mill position at the time it is clicked rather than completing the homing procedure.

NOTE: If desired, use this procedure in conjunction with one or more dial indicators mounted at the ends of mill travel to provide a more accurate method of manually referencing the mill.

6.6.4 Limiting a G30/M998 Move

Select *G30/M998 Move in Z Only* to move to the tool change position in the Z-axis only when using a G30 or M998 command. Otherwise, a coordinated X, Y, Z move occurs on G30 or M998.

The G30 or M998 G-code commands can be used to move the mill to a pre-set position. The position is settable using the *Set G30* button on the *Offsets* screen. A G30 or M998 command is typically programmed right before a tool change line in G-code program files so that the spindle head clears the workpiece with sufficient distance to be able to change tools. For more information on using a G30 or M998 command, refer to chapter 7, *Programming*.

6.6.5 Enabling Feeds and Speeds Suggestions in Conversational Programming

Select *Conversational Feeds and Speeds* to enable feeds and speeds suggestions in PathPilot when using conversational programming. For more information on use, refer to *Using Feeds and Speeds Suggestions* later in this chapter.

6.6.6 Enabling Accessories

Use the *Settings* tab to enable optional accessories that work with PathPilot.

6.6.6.1 Enabling CNC Scanner

Select *Enable CNC Scanner* if you are using an optional CNC Scanner. The *Scanner* tab displays. For information on installation and use, refer to the documentation that ships with the product.

6.6.6.2 Enabling Soft Keyboard

Select *Enable Soft Keyboard* to provide a soft keyboard for use with the optional Touch Screen LCD Monitor (PN 35575). For information on initial setup and calibration of the touch screen, refer to documentation that ships with the product.

When you select a DRO field, a numeric keypad opens on screen (see **Figure 6.17**). When you select either the MDI line, the *Save* field, the *Save As* field, or the conversational title DRO, a QWERTY keyboard opens on screen (see **Figure 6.18**). Press *Enter* to accept the value typed; press *ESC* to exit the soft keyboard and restore the previous value.

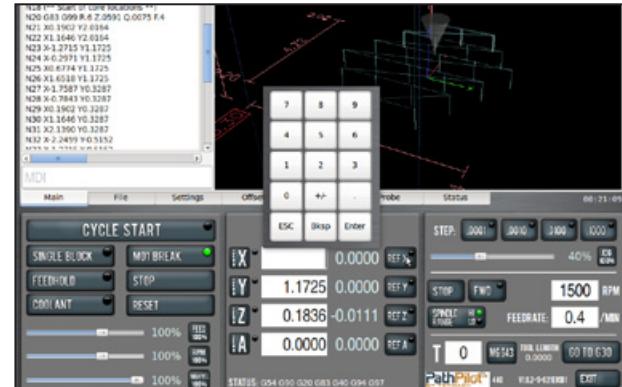


Figure 6.17

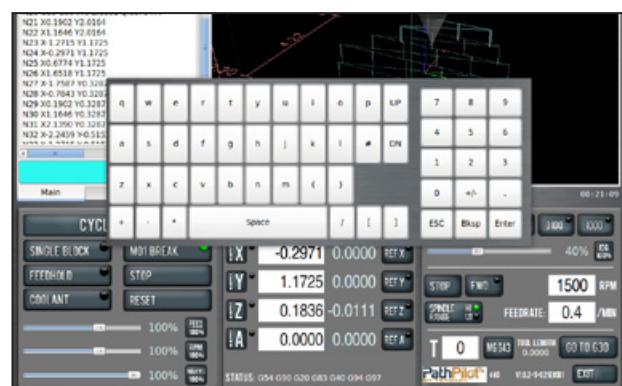


Figure 6.18

PATHPILOT INTERFACE

6.6.6.3 Enabling USB I/O Board

Select *Enable USB IO Board* if you are using an optional USB M-code I/O Interface Kit (PN 32616). For information on installation and use, refer to the documentation that ships with the product.

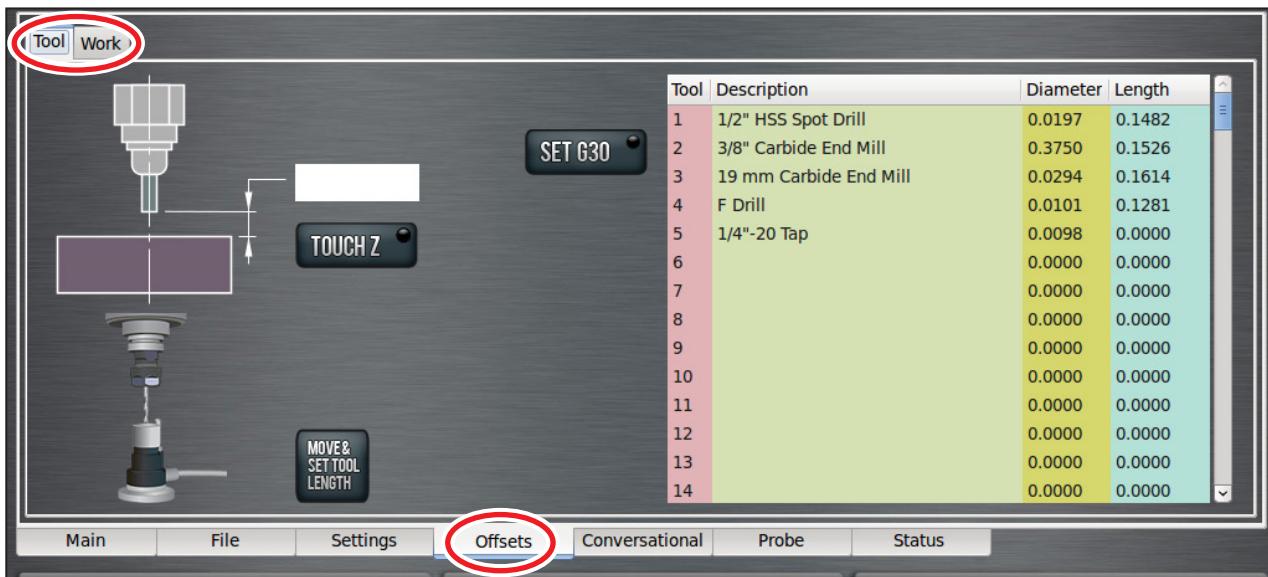


Figure 6.19

6.7 Offsets Tab

The *Offsets* tab reveals two sub tabs, *Tool* and *Work* (see **Figure 6.19**).

Work offsets are a concept that allow the operator to think in terms of X/Y/Z coordinates with respect to the part instead of thinking of them with respect to the mill position. Up to nine work offsets can be saved in the control. These are selected either by typing G54, G55, ... (up to G59, G59.1, G59.2, G59.3) into the MDI line, or by including them in a G-code program.

Tool offsets allow the operator to use tools of different lengths or (when using G41/G42 cutter radius compensation) different diameters, while still programming with respect to the workpiece. The most common tool offset is the tool length offset, which is applied by the G43 command.

6.7.1 Tool Tab

The *Tool* tab displays a tool table on the right with fields available to input tool information (see **Figure 6.19**).

For a given machining operation, the X and Y position of the workpiece are fixed. Using multiple tools may introduce a situation with tools of differing lengths; in this case, the you will need to change the Z offset for each particular tool. PathPilot allows you to switch tools quickly, without the need to set up the mill every time a tool is mounted. Each tool and holder only needs to be measured once, either offline or in the mill.

6.7.1.1 Tool Measuring Techniques

Different tool measuring techniques may be used, but the three most common methods are:

- Offline measurement with a height gauge
- Automated measurement with an electronic tool setter
- Touching off tool to a reference surface

Offline Measurement with Height Gauge

The Tormach Tool Assistant Set (PN 31988) includes an 8" digital height gauge and a USB interface cable with touch trigger (see **Figure 6.20**).

1. Connect USB interface cable to any available USB port on the controller.
2. Move the digital height gauge to a block and press the Zero button on the touch trigger. The device is zeroed.

NOTE: Granite Surface Plate with Integrated Tool Hole (PN 31713), as shown in **Figure 6.20**, helps accurately measure tool length.

3. To measure tool height, mount the tool in a TTS tool holder and place on a block (see **Figure 6.20**). The tool height measurement is the distance from the block surface to the end of the tool tip.
4. Click on the appropriate line in the tool table (see **Figure 6.19**); the line is highlighted.
5. Press *Data* button on the USB interface cable to transfer measurement information to the *Length* field in the tool table (see **Figure 6.19**).

Automated Measurement with an Electronic Tool Setter

Setup: Before using an electronic tool setter, set the work offset such that the surface upon which the electronic tool setter sits is Z zero. A quick way to do this is to use the *Move And Set Work Offset* button on the *Offsets* screen's *Work* tab, with either a tool of known length in the spindle or with no tool in the spindle (and Tool Zero in the tool DRO). By doing this setup step, you are measuring the true tool lengths, and can interchange tools measured in the height gauge with tools measured with the electronic tool setter. After completing the setup, measuring a tool with the electronic tool setter is easy. Put the tool in the spindle, tell the control which tool is in the mill (type the tool number in the tool DRO), and then, with the tool centered over the electronic tool setter, click the *Move And Set Tool Length* button on the *Offsets* screen's *Tool* tab.

Measurement by Touching Off Tool

Refer to chapter 5, *Intro to PathPilot*. It is not as accurate, but works in many situations.

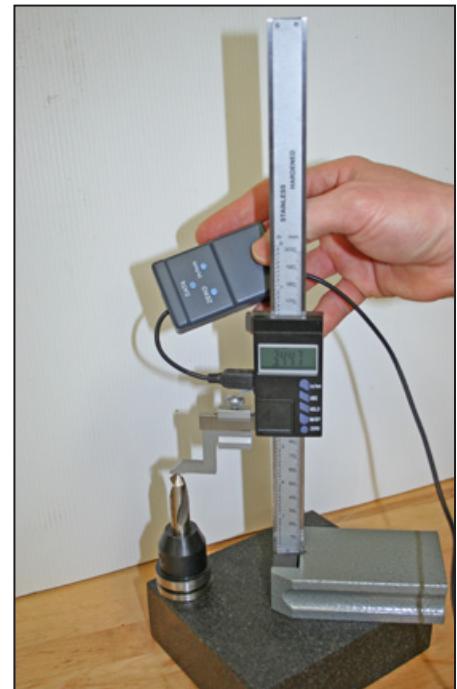


Figure 6.20

PATHPILOT INTERFACE

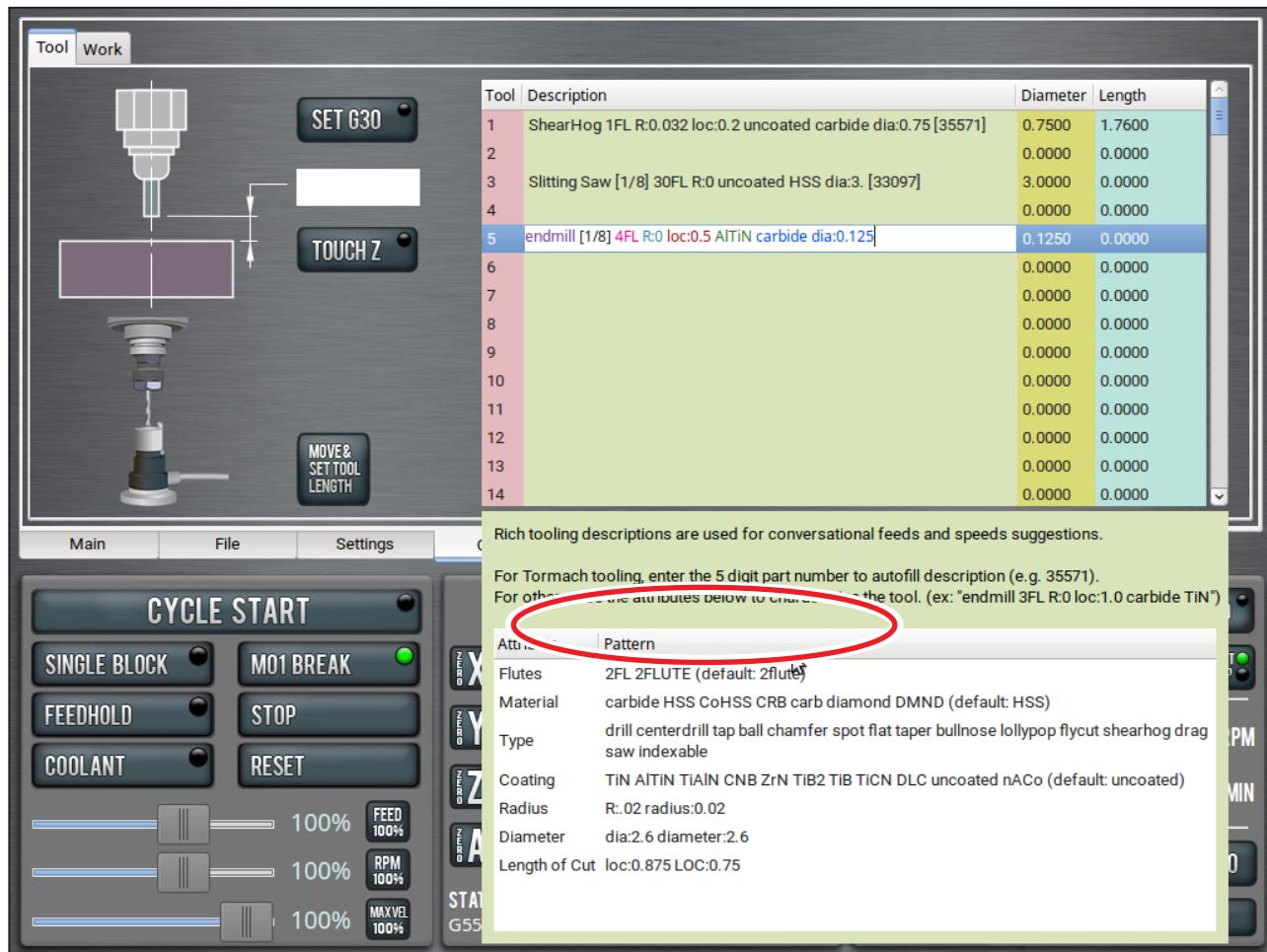


Figure 6.21

6.7.1.2 Creating Tool Descriptions

PathPilot uses keywords and patterns in the tool description to recognize tooling features. Refer to the section *Using Tool Keywords* later in this chapter for more information.

Example: "Dia:.3125 4FL R:03 AlTiN CRB variable loc:.75"

This description provides the following information for PathPilot to calculate machining information:

- 0.3125 tool diameter
- Four flutes
- 0.03 radius, or "bullnose"
- Aluminum-titanium nitride coating
- Carbide
- Variable helix
- 0.75 length of cut (loc)

To get accurate machining information, all tooling must be described with detail: the more detail, the better the results. Using a personal description likely won't contain meaningful information for PathPilot.

Example: "Gold colored end mill from middle drawer"

This description provides virtually no information. PathPilot defaults to basic cutter features about this tool:

- Two flutes
- Uncoated, high-speed steel end mill
- Length of cut based on the diameter

Manually Entering Tool Descriptions

You can manually enter tool descriptions in the *Tool Table* window. Descriptions are not case sensitive.

If a pattern or word in the description is recognized, PathPilot uses syntax highlighting to indicate a valid description (see **Figure 6.21**).

Refer to the section *Using Tool Keywords* later in this chapter for more information on tool description keywords.

PATHPILOT INTERFACE

Using Tool Keywords

Item	Pattern	Example	Notes
type	drill, centerdrill, tap, ball, chamfer, spot, flat, taper, bullnose, lollypop, flycut, shearhog, drag, saw, indexable	<ul style="list-style-type: none"> • Drill • Ball • Flycut • Drag 	"Drag" indicates that the tool is a drag tool, and has no (0) RPM associated with it.
flutes	A number followed by "FL" or "FLUTE"	<ul style="list-style-type: none"> • 4FL • 12FL • 2flute 	No flutes is specified the same as two flutes.
length of cut (or flute length)	"loc" followed by a colon, followed by a decimal number	<ul style="list-style-type: none"> • loc:0.875 	If no length of cut is specified, a length is assumed based on cutter diameter.
tool coating	TiN, AlTiN, TiAlN, CNB, ZrN, TiB2, TiB, TiCN, DLC, uncoated, nACo	<ul style="list-style-type: none"> • TiN • ZrN • TiB2 	No coating is specified same as "uncoated."
tool diameter	"diameter" or "dia" followed by a colon, followed by a decimal number	<ul style="list-style-type: none"> • Diameter:.0341 • dia:.750 	—
tool material	carbide, HSS, CoHSS, CRB, carb, diamond, DMND	<ul style="list-style-type: none"> • HSS • CoHSS • crb 	No tool material is specified the same as HSS (high-speed steel).
tool radius	"R" or "radius" followed by a colon, followed by a decimal number	<ul style="list-style-type: none"> • R:.02 • radius:0.02 	No radius is specified the same as a zero radius.

PATHPILOT INTERFACE

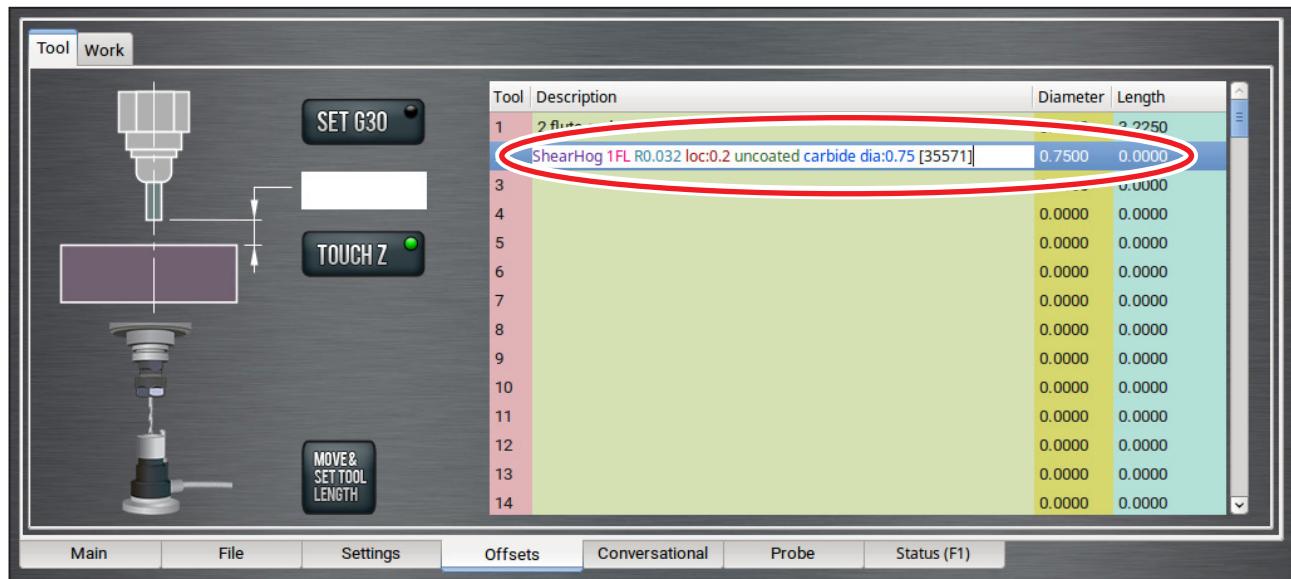


Figure 6.22

Generating Automatic Tool Descriptions

If you are using a Tormach tool, you can enter the part number to automatically generate tool descriptions in the *Tool Table* window (see **Figure 6.22**).

NOTE: If you're unsure of the part number, you can search for the tool at tormach.com.

1. From the PathPilot interface, on the *Offsets* tab, in the *Tool Table* window, select a blank line.
2. Type the part number for the tool.

Example: 35571

The full description and tool diameter for a ShearHog (PN 35571) displays.

3. You must enter the value for the *Length*.

6.7.2 Offsets Table

The offsets table on the *Offsets* screen (see **Figure 6.19**) displays an editable table of tool offsets (both geometry offsets and wear offsets) as well as a read-only table of work offsets. To alter a value in the tool offsets table, double-click on the field. After entering the desired new value, click *Enter* to accept the value. The mill must be powered on and not in reset mode to edit these fields.

6.7.3 Tool Offset and Fixture Information Backup

Make a periodic backup of the tool offset and fixture information and machine settings to store externally should the controller get replaced or need to be restored to factory settings.

Create a tool offset and fixture information backup on a PathPilot controller as follows:

1. Insert a USB drive into any open USB slot on the controller.

PATHPILOT INTERFACE

2. On the *Main* screen, type *ADMIN SETTINGS BACKUP* in the MDI line.
3. In the dialog box, navigate to a location to store the backup .zip file on the USB drive and rename if desired; click *Save*.

NOTE: Keep this file somewhere safe and easily accessible.

Restore tool offset and fixture information backup on a PathPilot controller as follows:

1. Transfer the tool offset and fixture information and machine settings backup to a USB drive; insert into any open USB slot on the controller.
2. On the *Main* screen, type *ADMIN SETTINGS RESTORE* in the MDI line.
3. In the dialog box, navigate to the backup .zip file on the USB drive; click *Open*. PathPilot exits, restores from the backup file, and then restarts PathPilot.

6.7.4 Work Tab

The *Work* tab (see **Figure 6.19**) displays a read-only table of work offsets. The active work offset is highlighted in this table.

NOTE: The table cannot be edited directly. To change the current work offset value, either type into an Axis DRO or use Zero button next to the Axis DRO.

6.8 Conversational Tab

The *Conversational* tab provides an interface for programming at the controller. Use *Conversational* to machine simple parts without the use of CAD/CAM.

6.8.1 Using Feeds and Speeds Suggestions

You can use PathPilot to automatically calculate feeds and speeds: from the *Conversational* tab, in the *Conversational DROs* group, select a material, a sub-type, and a tool (see **Figure 6.23**).

To calculate feeds and speeds, you must first make sure PathPilot has relevant details about the tooling. For more information, go to the section *Creating Tool Descriptions* earlier in this chapter.

1. From the PathPilot interface, on the *Conversational* tab, locate the *Material* dropdowns in the *Conversational DROs* group.
2. From the *Material* dropdown, select your material.

Examples:

- Aluminum
- Plastic

3. If required, from the *Sub-type* dropdown, select the material sub-type.

Examples:

PATHPILOT INTERFACE

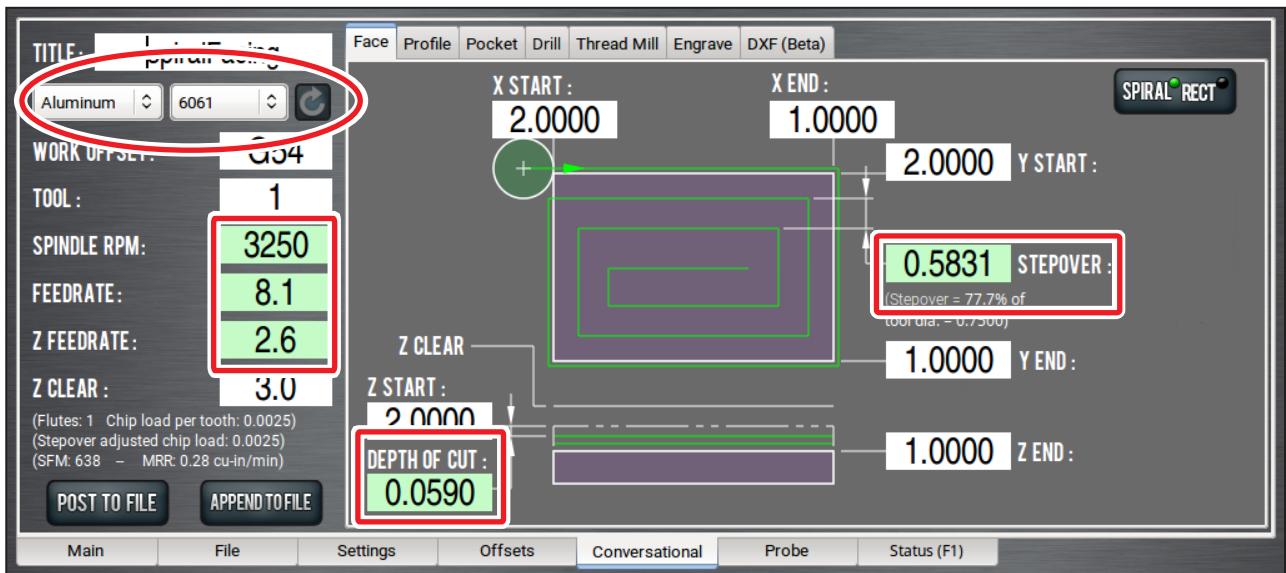


Figure 6.23

- -any-
 - 6061
4. In the *Tool* DRO, type the assigned tool number.
 5. Click *Refresh* (to the right of the *Sub-type* dropdown).

The following machining-related DROs are calculated:

- *Spindle RPM*
- *Feedrate*
- *Z Feedrate*
- *Depth of Cut* (if milling)
- *Stepover* (if milling)
- *Peck* (if drilling)

NOTE: After PathPilot calculates values for the machining-related DROs, the background turns green (see Figure 6.23).

PATHPILOT INTERFACE

6.8.1.1 Adjusting DRO Values

After selecting the material and tool, you can adjust the values in the calculated DROs, like *Feedrate* or *Stepover*. Adjusting the value in one of these DROs does not change the value in the other machining-related DROs.

Once you adjust the value in the DRO, the background switches from green back to white (see **Figure 6.24**). This helps you identify which DROs have suggested values (those with a green background), and which DROs have values you've supplied (white background).

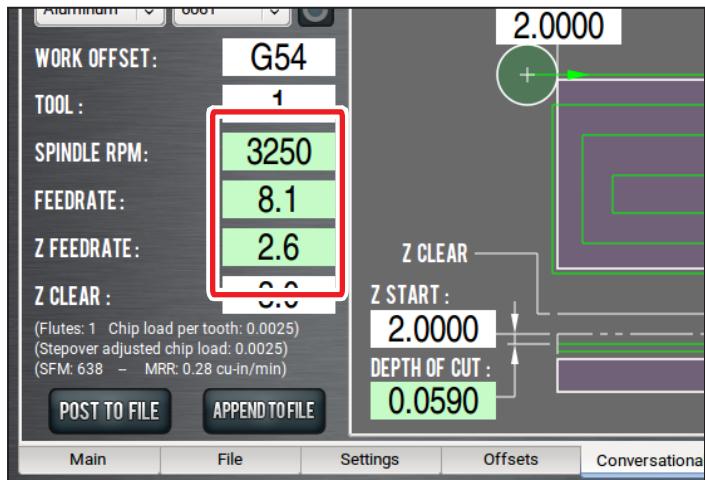


Figure 6.24

6.8.1.2 Resetting DRO Values

The suggested feeds and speeds are no longer valid if:

- You select different material or sub-type values, or if you type a new value in to the *Tool* DRO.

The suggested feeds and speeds are made by taking into account all of these values. Changing any value requires you to refresh.

- You select a different *Conversational* tab.

The suggested feeds and speeds are made by taking into account the current, specific conversational operation — like Face, or Pocket. Changing your conversational operation requires you to refresh.

When the feeds and speeds are no longer valid, the *Refresh* button turns green, and the machining-related DRO backgrounds switch from green to white (see **Figure 6.25**).

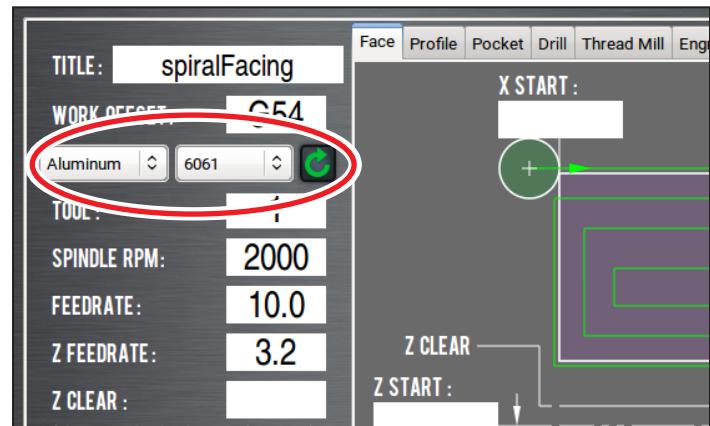


Figure 6.25

6.8.1.3 Using Chip Load Information

On the *Conversational* tab, in the *Conversational DROs* group, there are tips that are displayed based on the calculations that PathPilot is performing (see **Figure 6.26**):

- Chip load information

Chip load — the amount of material removed per tooth — is based on the number of flutes, RPM, and feedrate.

PATHPilot INTERFACE

Chip thinning takes the stepover (the horizontal depth of cut into the workpiece) into account, and provides the actual chip load.

As the stepover value decreases, the actual chip load decreases. If the stepover is too small, the cutter may not have enough contact with the material to cut — effectively resulting in pre-mature tool wear.

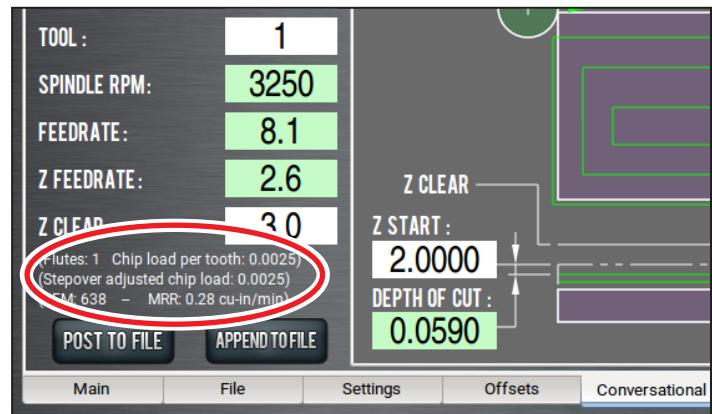


Figure 6.26

6.8.2 Face Tab

Face is generally used for cutting an accurate top surface from rough stock, cutting successive XY-planes over a Z range (see **Figure 6.27**).

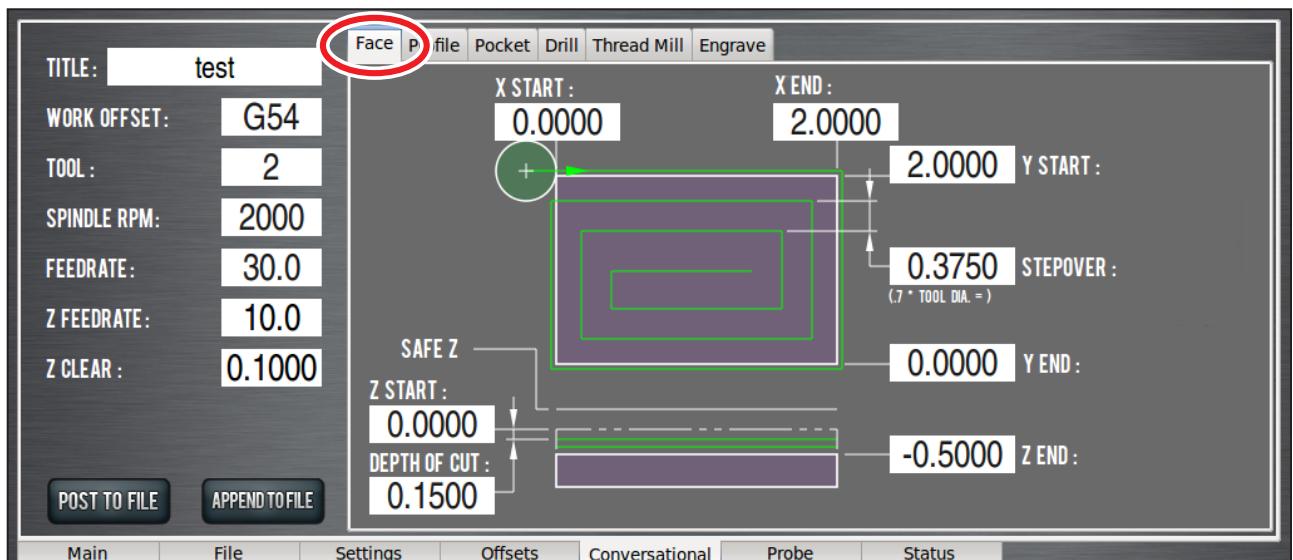


Figure 6.27

It is assumed that the top of the stock is free of any clamps or other work holding devices, such as when the stock is held in a vise. The start of each Z pass is intended to be off to the side of the workpiece then move in XY to start cutting at the X Start, Y Start corner. This avoids the need for plunging the Z Depth of Cut move into the workpiece.

PATHPILOT INTERFACE

Therefore, the area around this corner must be clear of obstructions down to *Z End*. The tool diameter also extends beyond the workpiece X and Y edges by an amount dependent on the tool diameter and the stepover values, so *Z End* must be above the vise jaws.

The G-code routine starts with a move to G30, which typically is the park, or tool change position. Next comes a tool change if needed, a rapid move in XY to the workpiece start, and a rapid down in Z to *Z Clear*. An XY pass starts with an adjusted *Z Depth of Cut*, then a rectangular spiral from the workpiece perimeter, ending at the center. If a finish pass with different parameters is needed, save the current file, edit the current screen to the finish configuration and append to the saved file.

XY DROs

Start and End – These DROs should be set to the location of the workpiece edges. Tool paths, such as a lead-in, that are normally outside of the workpiece area are set in reference to these values, so no adjusting beyond the actual location of these edges should be needed.

Stepover – This is the space between spiral tool paths. To prevent uncut areas in the spiral corners, the stepover value should be limited to 80 percent of the tool diameter (see **Figure 6.28**). A stepover of 0 may be entered which invokes a center only cut. This is more formally called by X or Y values that create a workpiece width less than 70 percent of the tool diameter.

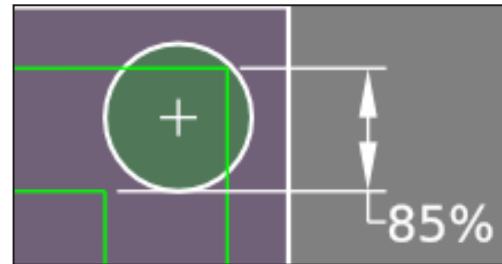


Figure 6.28

Z DROs

Z Start and End – The first Z pass will cut at *Z Start – Depth of Cut* adjusted. The last Z pass will cut at the *Z End* location. For a single Z pass at *Z End*, enter 0 or a full Z range value into the Depth of Cut DRO.

Depth of Cut – The *Depth of Cut* entered into the DRO is later adjusted within the Z range, *Z End – Z Start*, so each Z pass has the same depth instead of having a short depth on the last pass. For a single pass at *Z End* enter 0 or a full Z range value into the *Depth of Cut* DRO.

Z Clear – This is the Z location the tool moves or retracts to when starting or ending a Z pass. This should be set to clear any obstructions in the path between the end of one Z pass and the beginning of the next.

6.8.3 Profile Tab

Profile cuts an XY area with successive Z *Depth of Cuts* to form a rectangular island (see **Figure 6.29**).

PATHPilot INTERFACE

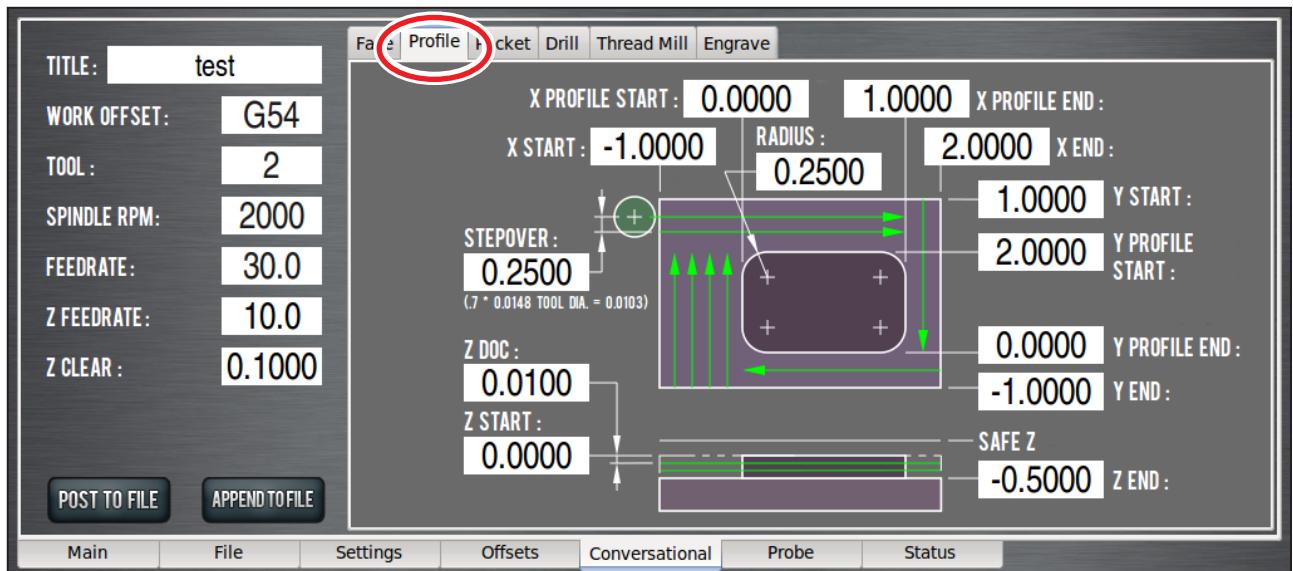


Figure 6.29

The outer bound of the area is the stock material's outer edges. The inner bound is the island perimeter. For the cutting routine, the area is divided into four sections (north, east, south, and west). As with *Face*, the starting position for cutting each section is off the workpiece with an X or Y feed into the workpiece, thus avoiding a Z plunge cut. Cutting paths are restricted to climb cutting, so the tool is retracted to *Z Clear* at the end of each sweep of a section, with a rapid move to the beginning of the section for the next sweep. After each section is cut, the corner radii, if any, are cut with a tool path that travels around the perimeter of the island. This process is repeated for each *Z Depth of Cut* pass. If a finish pass is needed, leave enough material, then append your finishing G-code (usually a single pass around the perimeter) to this file later. Feed rate on the radius cuts are adjusted to compensate for the difference between the tool control point rate (at the tool center) and the actual rate at the radius surface.

X and Y Start and End DROs

Start and End – These DROs should be set to the location of the workpiece edges. Tool paths outside of the workpiece area are set in reference to these values so no adjusting beyond the actual location of these edges should be needed.

Profile Start and End – The tool radius is used to create the tool path, so these DROs should be set to the location of the profile outer edges.

Radius – Enter 0 if no corner radius is desired. Valid radii values are from 0 to one half of the island's narrow width (or limited to full radii on the long ends of the island).

Stepover – This is the tool path offset between section sweeps. A stepover of 0 creates a single pass (or rectangular slot) around the perimeter (outside) of the boss.

PATHPILOT INTERFACE

Z Start and End – The first Z pass will cut at Z Start – Depth of Cut adjusted. The last Z pass will cut at the Z End location. For a single Z pass at Z End, enter 0 or a full Z range value into the Depth of Cut DRO.

Depth of Cut – The *Depth of Cut* entered into the DRO is later adjusted to fit evenly within the Z range (*Z End – Z Start*), so each Z pass has the same depth instead of having a short depth on the last pass. For a single Z pass at Z End, enter 0 or a full Z range value into the *Depth of Cut* DRO.

Z Clear – The Z location the tool moves or retracts to when starting or ending a Z pass, a section sweep, or a section change. This should be set to clear any obstructions between path changes.

6.8.4 Pocket Tab

Pocket cuts a rectangular or circular pocket (cavity). The rectangular pocket can have a corner radius specified, or otherwise the tool paths have sharp corners (see **Figure 6.30**).

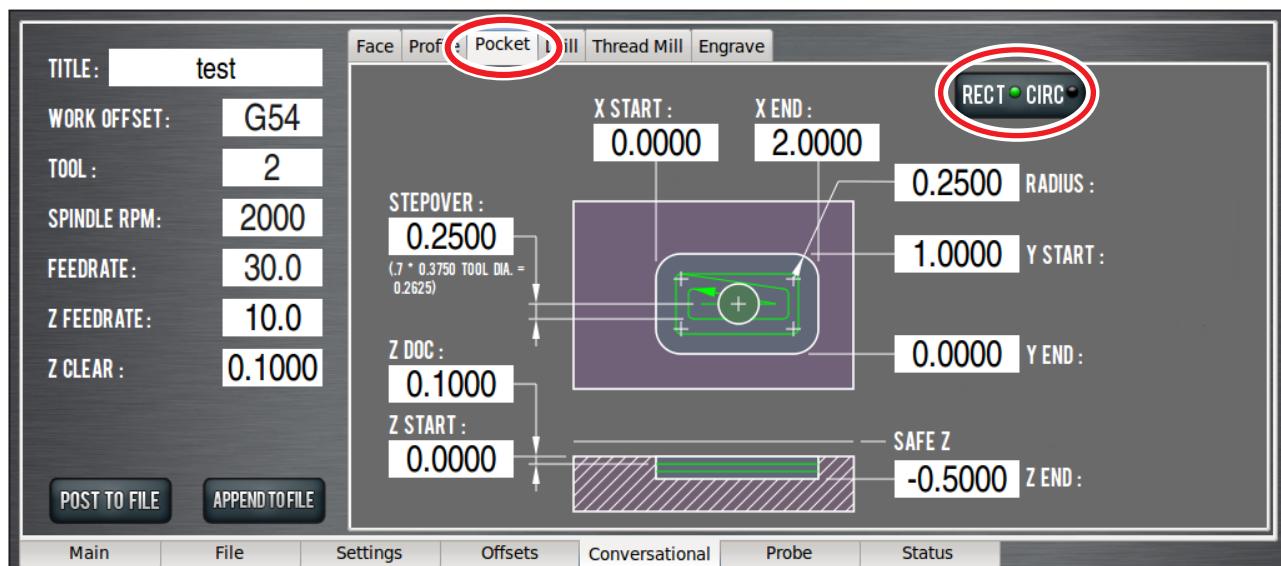


Figure 6.30

6.8.4.1 Rectangular

The general tool path pattern for Pocket-Rectangular depends on the size of the width (width being considered the smaller of X or Y widths) and length of the pocket relative to tool diameter. The pattern within each Z *Depth of Cut* pass is repeated within the Z range (*Z End – Z Start*), but the entry and clearing patterns may be different. There are three sub-patterns: entry, clear (out material), and perimeter.

If tool diameter is bigger than pocket width, no G-code is produced and an error appears on the *Status* screen.

If the tool just fits within the pocket width and length, a straight Z plunge in the pocket center is used, therefore a center cutting end mill is needed. Next comes a single pass around the perimeter. This is repeated for each Z *Depth of Cut* pass.

If the tool just fits within the pocket width, but length is greater than 2x tool diameter, this allows a linear ramp entry which also does the material clearing.

The linear ramp is limited to a Z slope of 2° or less (the angle is adjusted smaller to fit the slot length). A single perimeter cut is done next.

If the pocket width and length are greater than 2x tool diameter, this allows a helical entry which cuts a hole of 2x tool diameter in the center of the pocket. Material clearing is done by squaring up the hole, then cutting wings to each side of the pocket length. Finally, a perimeter cut is done.

X and Y Start and End DROs

Start and End – These DROs should be set to the location of the pocket edges.

Radius – Enter 0 if no corner radius is desired. Valid radii values are from 0 up to one half of the pocket's narrow width (or limited to full radii on the long ends of the pocket). Of course, the actual corner radii must be larger than or equal to the tool's radius, but *Pocket* tolerates radius entries less than the tool radius – the tool path will just be a sharp corner.

Stepover – This is the offset between adjacent tool paths. A stepover of 0 creates a single pass (or rectangular slot) around the perimeter (inside) of the pocket.

Z DROs

Z Start and End – The first Z pass will cut at *Z Start – Depth of Cut* adjusted. The last Z pass cuts at the *Z End* location. For a single Z pass at Z End, enter 0 or a full Z range value into the *Depth of Cut* DRO.

Depth of Cut – The *Depth of Cut* entered into the DRO is later adjusted to fit evenly within the Z range (*Z End – Z Start*), so each Z pass has the same depth instead of having a short depth on the last pass. For a single Z pass at Z End, enter 0 or a full Z range value into the *Depth of Cut* DRO.

Z Clear – The Z location the tool moves or retracts to when starting or ending the *Pocket* routine.

6.8.4.2 Circular

Pocket-Circular has a different entry for cutting a circular pocket dependent on the pocket diameter and the tool diameter (see **Figure 6.31**).

If the tool diameter is bigger than pocket diameter, no G-code is produced and an error appears.

If the tool just fits within the pocket diameter, a straight Z plunge in the pocket center is used, therefore a center cutting end mill is needed. Next comes a single pass around the perimeter. This is repeated for each Z *Depth of Cut* pass.

PATHPILOT INTERFACE

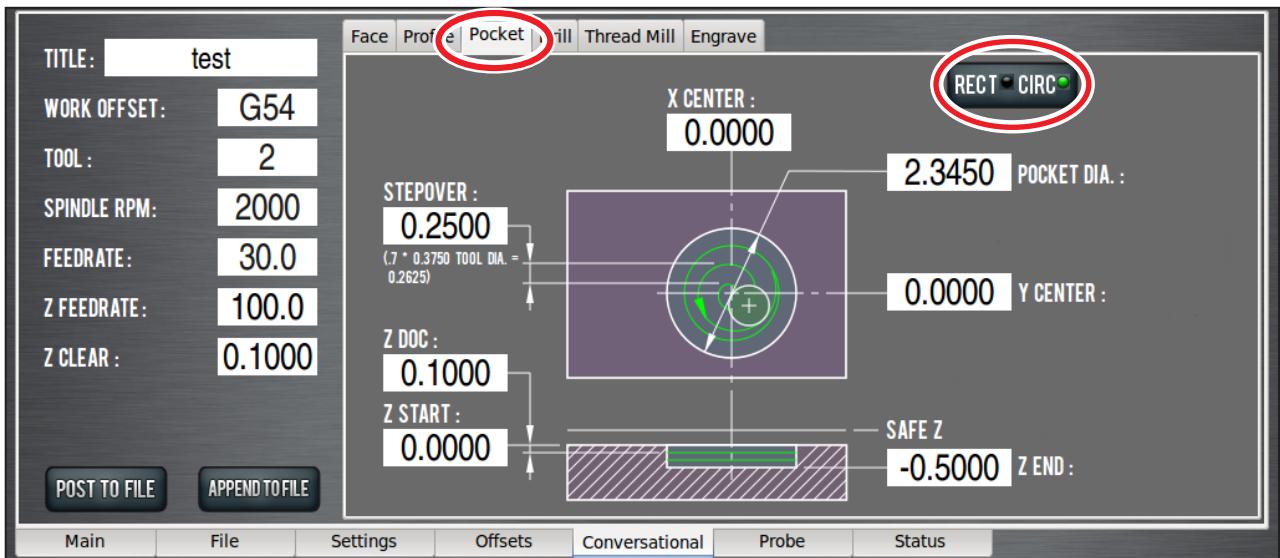


Figure 6.31

If the pocket diameter is greater than 2x tool diameter, this allows a helical entry which cuts a hole of 2 x tool diameter in center of pocket. Material clearing is done with a spiral cut out to the pocket diameter, plus a cut around the perimeter.

XY DROs

X and Y Center – These DROs should be set to the location of the pocket center.

Pocket Dia. – Enter value of the pocket diameter. The tool radius is used to set the tool path diameter.

Stepover – This is the tool path offset between each rotation of the spiral cut. A stepover of 0 creates a single pass (or circular slot) around the perimeter (inside) of the pocket.

Z DROs

Z Start and End – The first Z pass cuts at *Z Start – Depth of Cut* adjusted; last Z pass cuts at the *Z End* location. For a single Z pass at *Z End*, enter 0 or a full Z range value into the Depth of Cut DRO.

Depth of Cut – The *Depth of Cut* entered into the DRO is later adjusted to fit evenly within the Z range (*Z End – Z Start*), so each Z pass has the same depth instead of having a short depth on the last pass. For a single Z pass at *Z End*, enter 0 or a full Z range value into the Depth of Cut DRO.

Z Clear – The Z location the tool moves or retracts to when starting or ending the Pocket routine.

PATHPilot INTERFACE

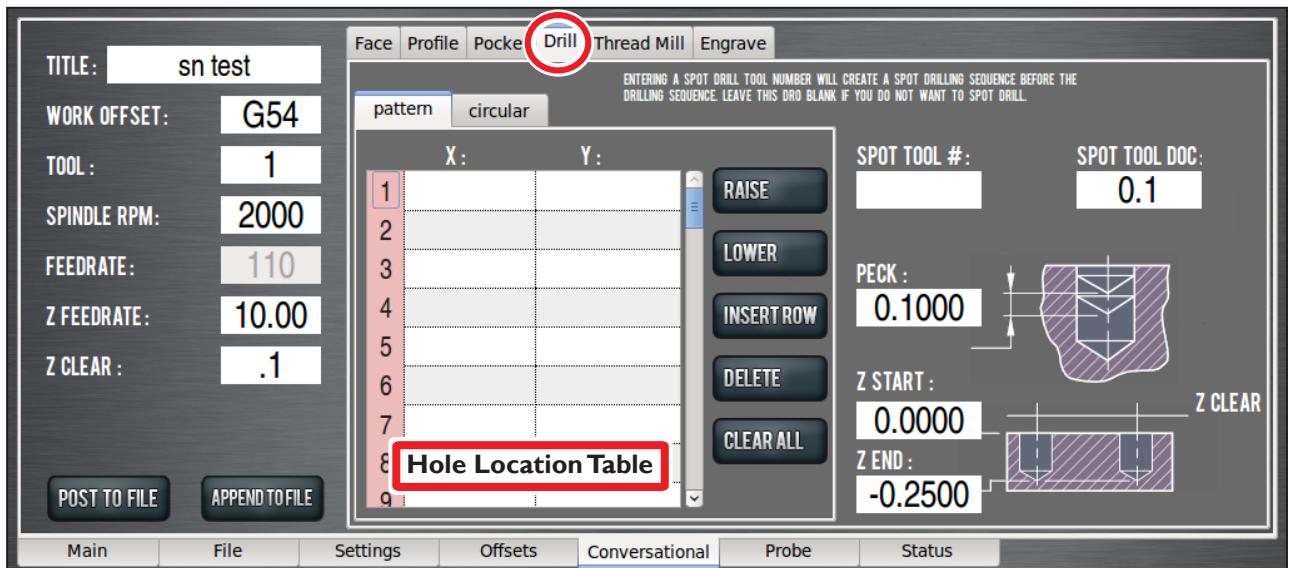


Figure 6.32

6.8.5 Drill Tab

Drill provides a means to create a hole location list, then, based on DRO entries, configures an appropriate canned cycle G-code to create holes – either *Pattern* or *Circular* (see **Figure 6.32**).

Hole Location Table – Used for making a list of X and Y locations for each hole using the same tool, Z and common DRO entries (see **Figure 6.32**). To create holes using different tools or other parameters, post the first group, clear the table, enter the next group of locations and other parameters, then append the new list to the existing posted file.

Holes in a list are completed in order from top to bottom. Rearrange row order by using the *Raise* and *Lower* buttons. To move a row, first activate it by clicking anywhere on the desired row, which highlights it (in blue) then select the *Raise* and *Lower* button. To edit an X or Y cell, click on the desired row, then click the desired cell. An active cell shows up as a white box with a cursor marker (which looks like |) on a blue row. If there is already a number in the selected cell, it is blocked in blue and is replaced with any number typed in. To edit an existing number, be sure to click on the number until a cursor appears. The *Clear All* button clears all entries in the table. Upon leaving a cell, the entry is checked to see if the entry is a valid number. If not, the entry is erased and an error shows up in the *Status* tab.

Rows are checked when the *Post* button is clicked. Any missing entries (an X without a Y, a Y without an X, or an empty row before the last row with an entry) stop the posting and insert the text “??” in the cells with missing entries. Just fill in the missing entries, delete any “??” entries, as well as *Lower* any empty rows past the last row with an entry, then try posting again.

Drill – Uses one of the canned G8x cycles to drill a hole at each location called out in the hole table (see **Figure 6.32**). The drill cycles available are: G81 Drill, G82 Drill with Dwell, and G83 Drill with Peck (features can not be combined, peck cancels dwell).

PATHPILOT INTERFACE

Since it is usually more convenient to touch-off a drill on its point, that configuration is presented in the graphics. Hole depth is usually defined as the full diameter portion of the hole, so the Z length from the drill point to the corner may need to be considered.

Dwell – An entry greater than 0 replaces G81 with G82 in the G-code, unless there is an entry greater than 0 in Peck. The G82 routine feeds at the Z Feedrate (a DRO in the left panel) until reaching the bottom of the hole, then the position is maintained during the period set by *Dwell*. This is usually used to let the tool complete the cutting of the hole bottom before retracting. A revolution calculation is presented in the graphics to aid in setting an appropriate dwell value (such as half revolution for a two flute drill).

Peck – An entry greater than 0 replaces G81 with G83 in the G-code. The G83 routine feeds at the Z Feedrate starting from *Z Clear* down a Peck distance, then rapid retracts to *Z Clear*, and rapid returns to start the next peck. The peck distance is not adjusted so the first and last peck will likely be shorter than the PECK setting.

Z DROs

Z Start and End – G8x starts at the *Z Clear* location and ends at *Z End* location.

Z Clear – This is the Z location the tool moves or retracts to at the start, end, and while pecking, as well as moving between holes, so it must clear any obstructions along the path between holes.

The *Circular* tab creates a specific hole pattern of evenly spaced holes around a circumference, also known as a bolt pattern (see **Figure 6.27**). As with the *Pattern* tab, all features and corresponding DROs, like *Spot* and *Peck*, are retained.

Circular DROs

Number of Holes – Specifies the number of holes in the pattern. This must be greater than zero.

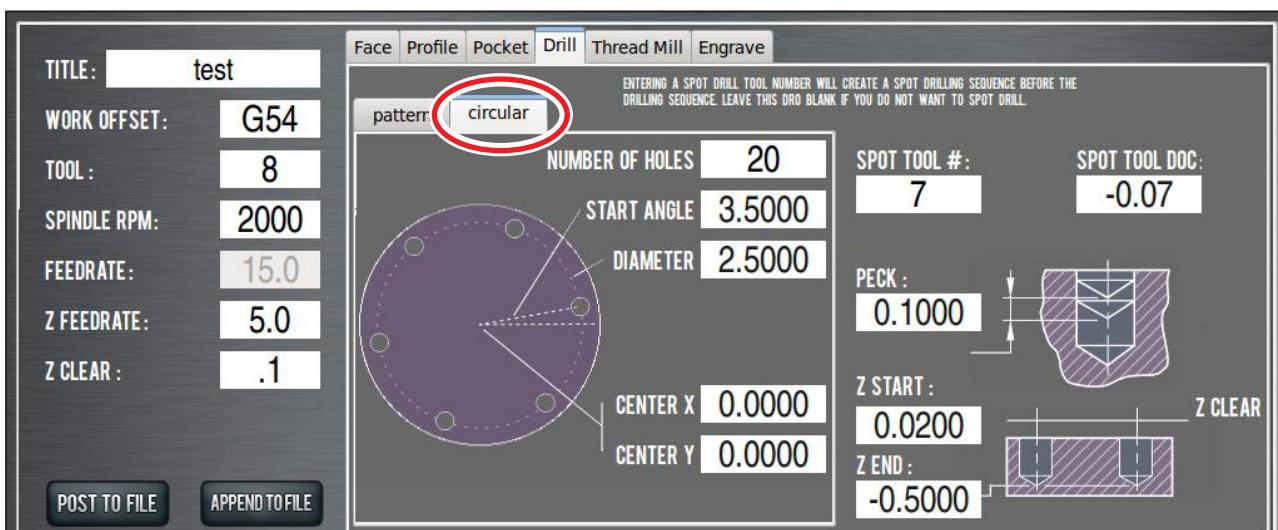


Figure 6.33

PATHPilot INTERFACE

Start Angle – Specifies the angle from angle 0. Angle 0 is a base (horizontal) line from the center point going right (east) to the circumference. The angle from the base line can be either positive or negative, up to 90 degrees (or -90 degrees) and rotates the pattern either clockwise or counterclockwise. A negative angle produces a clockwise rotation; a positive angle produces a counterclockwise rotation. For example, to create a hex pattern with flats on the top and bottom, enter 0 into the *Start Angle* DRO. To create a hex pattern with flats on the left and right sides, enter 30 (or -30) into the *Start Angle* DRO.

Diameter – The size of the circular pattern as defined by a line through the center point of each hole.

Center X, Center Y – Defines the center point of the circular pattern.

Spot Tool # – If this DRO contains a valid tool number when *Post To File* is clicked, a spot drilling sequence using this tool number will occur prior to the drilling sequence. The *Feedrate*, *Spindle RPM*, and *Z Clear* from the drilling sequence will be used for the spot drilling operation. The depth of cut for the spot drilling will be taken from the *Spot Drill DOC* DRO.

Spot Tool DOC – If the drilling operation includes spot drilling, this DRO will be used to determine the depth of cut for the spot drilling operation (for more information on defining a tool for a spot drilling operation, refer to *Spot Tool #* earlier in this section).

6.8.6 Thread Mill Tab

The thread milling routine produces helical tool paths needed for milling straight external or internal right-handed threads based on pitch, diameter, and length (see **Figure 6.34** and **Figure 6.35**).

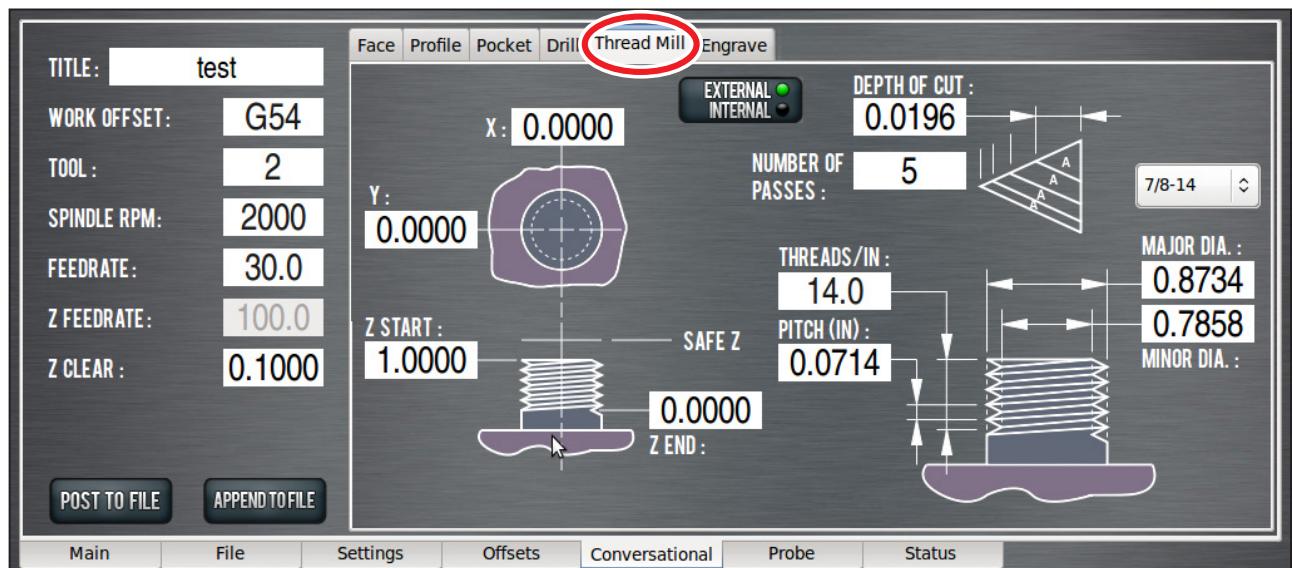


Figure 6.34

PATHPILOT INTERFACE

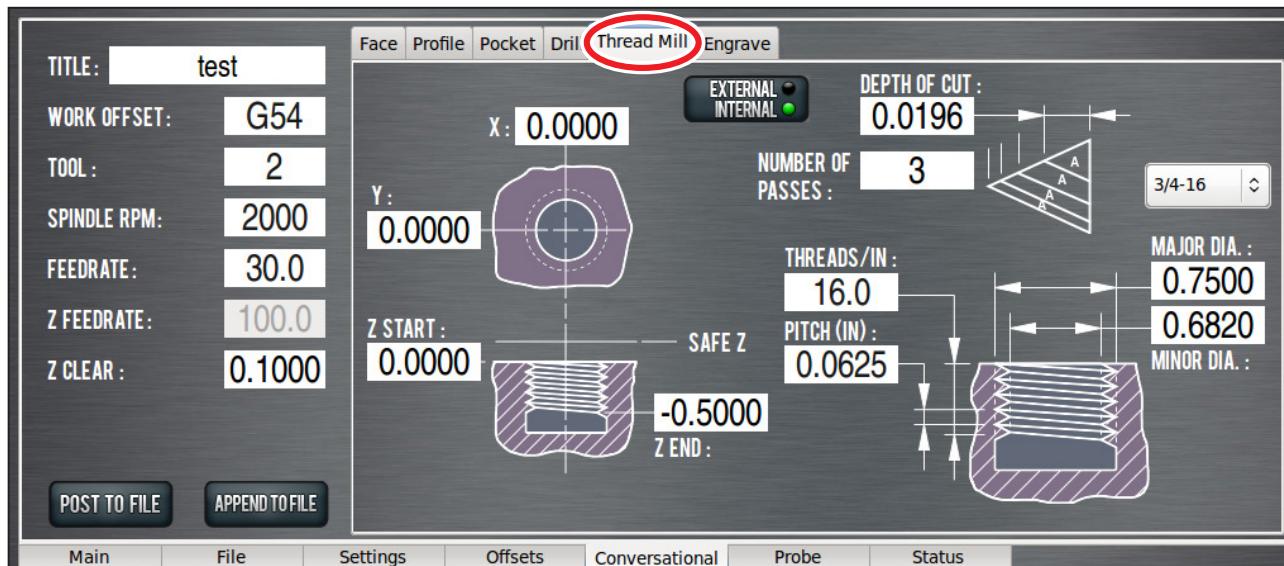


Figure 6.35

Thread Table – Contains values for some common threads. The threads listed follow the current unit setting (inch or millimeter). Once a selection is made, the data from the selected thread is copied to the appropriate DROs. This table is stored in user-editable text files found in the *thread_data* subdirectory of the G-code folder on the controller's hard drive; to edit (e.g., to add to or modify the defaults), highlight the file and click *Edit G-code*. For more information on files stored on the controller's hard drive, refer to *File Tab* section earlier in this chapter.

NOTE: The values entered in these tables assume a full form thread tool. If using a fine point threading tool to cut coarse threads, the root diameter must be modified to account for the smaller tool nose radius of the fine point threading tool.

XY DROs

X and Y – These DROs locate the center of the threaded stud or hole.

Major and Minor Diameter – Sets the start and end diameter of the thread peak and valley.

Depth of Cut – Sets the amount of material cut in each helical pass. The value entered is the distance (change in radius) the tool is fed on the first pass. This first pass cuts a triangular area which is related to the chip load. Subsequent cut depths are set to cut the same amount of area, so the linear feed gets smaller for each pass. The tool is also fed in on a *compound* angle of 30°, keeping the cuts to one face of the tool. The number of passes that fit in a thread depth is calculated and presented in the *Number of Passes* DRO.

Number of Passes – This DRO value is either calculated from the *Depth of Cut* value or can be entered here, which invokes (upon pressing the *Enter* key) a calculation and entry to the *Depth of Cut* DRO.

Z DROs

Z Start and End – Sets the location of the thread start and end. The tool will actually go beyond *Z End* due to the cutting tip width and the Z component of the compound feed angle and thread depth.

Z Clear – This is the Z location the tool moves or retracts to when starting or ending a Z pass. This should be set to clear any obstructions in the path between the end of one Z pass and the beginning of the next.

Threads/Unit and Pitch – Pitch is used to set the helix feed in Z per turn. An entry in one of these DROs will invoke a calculation and entry into the other, so enter whichever type of setting handy.

6.8.7 Engrave Tab

The *Engrave* tab contains functions to engrave a single line of text cut in a single horizontal pass (along the X-axis). This is a basic text engraving routine best suited for engraving True Type stick or outline fonts into things like simple plaques, control panels, or data plates.

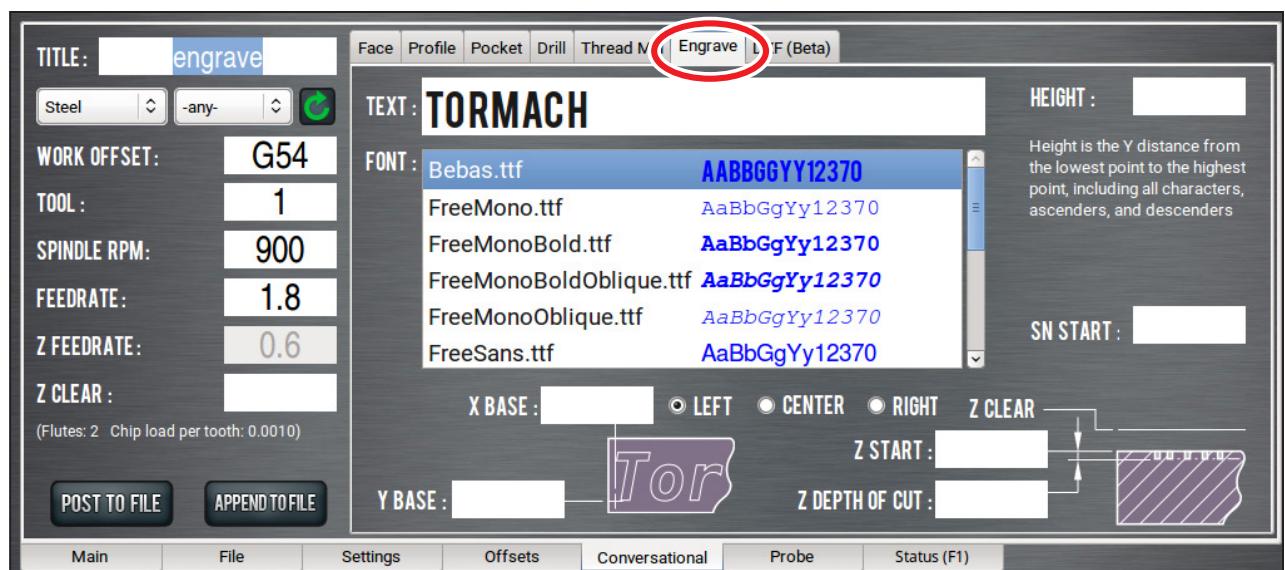


Figure 6.36

Fonts describe paths of the tool control point; therefore, the tool's effective cutting diameter may need to be considered for overall character size.

Serial numbers – a number that sequentially increases with each Cycle Start – can be engraved alone or added to the end of any desired text. Serial numbers use their own non-proportional font and are scaled to match the defined font extents. For more information on adding serial numbers to an engraving routine, refer to *SN Start* later in this section.

X and Y Base – Sets the location of the left side of the first character's baseline. If any characters in the text have descenders, such as *y* or *g*, they extend down below the baseline. The tool's effective cutting diameter may cut an area before or beyond the start location (see **Figure 6.37**).

PATHPILOT INTERFACE

Height – Sets the Y distance from bottom to top of text. This includes ascenders and descenders, but not the tool cutting diameter (subtract this diameter from the overall desired height to get a more accurate value to enter). Height is used with the font data to calculate a scale value that is applied to the character paths in the G-code. The actual height may vary and need adjustment.

Text DRO – Is the text to be engraved. A sample of the text in the selected font is updated when the *Enter* key is pressed.

SN Start – Sets a starting serial number. Add zeros in front of the first digit as a hint to the number of decimals to be engraved in the series (including leading zeros). For example, if '0012' is entered, '0012,' '0013,' '0014,' '...' will be engraved. If '99' is entered, '99,' '100,' '101,' '...' will be engraved. The current serial number is stored internally; to view, hover over the *SN Start* DRO.

NOTE: Leave the *Text DRO* blank to only engrave a set of sequential serial numbers; likewise, leave *SN Start* blank to only engrave a line of text. It is an error if both the *Text DRO* and *SN Start* are blank.

Font – Lists the True Type font files found in the font directory. Scroll through and click on the desired font; clicking presents a sample in the *Text DRO*. Some font files do not render in this box, but may be viewed by posting the file and checking the font in the *Main* tab's preview window (see **Figure 6.36**). True Type font files may be added to the *Font* list by transferring font files to the *gcode/engraving_fonts* sub-directory in the controller's home directory (for more information, refer to *File Tab* section earlier in this chapter). Power the controller off and back on to refresh new files in the *Font* list.

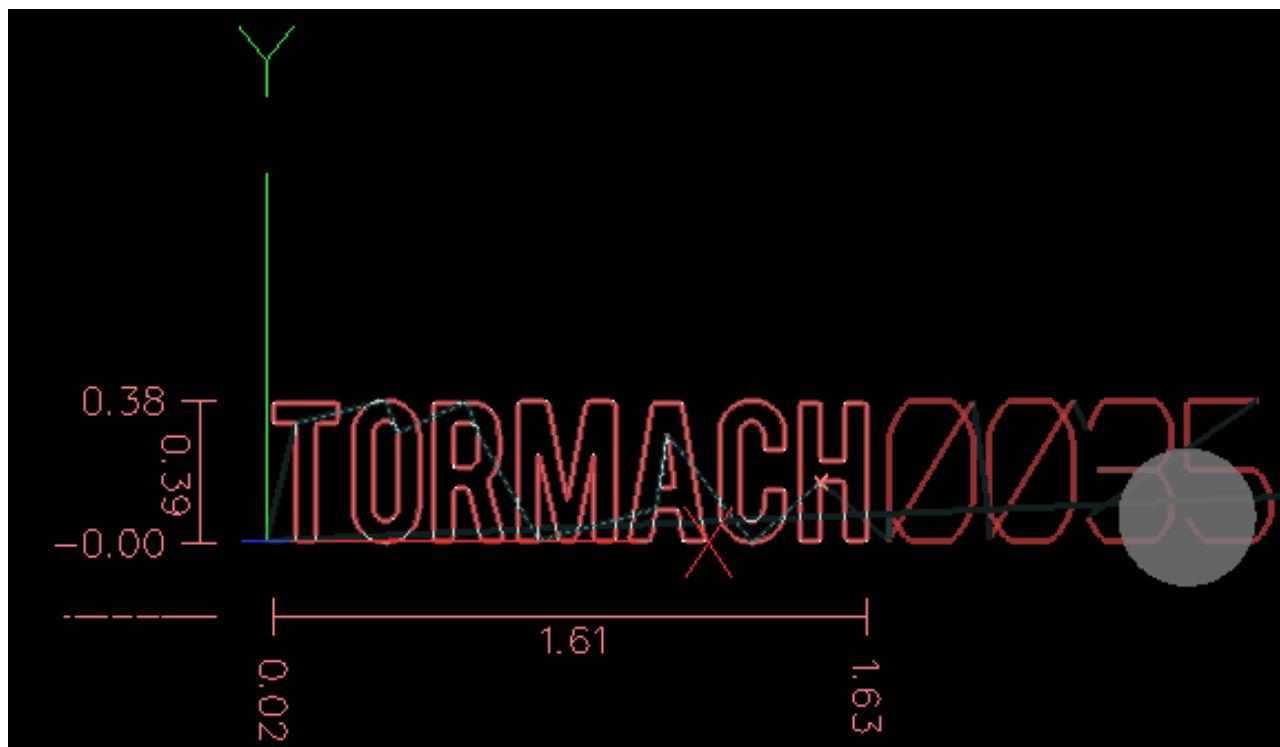


Figure 6.37

Z DROs

Z Start – Sets the location of the surface to engrave.

Z Depth of Cut – Is the depth the cutter is fed into the workpiece.

Z Clear – Is the Z location the tool moves or retracts to at the start and end of the engraving routine, and when moving between characters.

6.8.8 DXF Tab

You can import a .dxf file (Drawing Exchange Format) into PathPilot to generate G-code, which can then cut the shape (or shapes) described in the .dxf file. For example, you could use this feature to engrave logos or artwork.

1. Click the *File* DRO.

The *File Selector* dialog box opens.

2. Select the .dxf file, and then click *Open*.

The shapes from the selected file are loaded into the *Preview* window in the *DXF* tab.

NOTE: The .dxf file must already be transferred to the PathPilot controller.

3. Set the offsets: In the *X Offset* DRO and the *Y Offset* DRO, type the offset value added in the XY direction from the bottom left corner of the .dxf drawing.
4. In the *Scale* DRO, type the scale factor for the drawing. The value typed in the *Scale* DRO is used as a multiplier for the .dxf dimensions.

NOTE: The scale factor is applied to the entire drawing. For example, if you type 1.0 in the Scale DRO, the .dxf is scaled at 100 percent. If you type 2.0 in the Scale DRO, the .dxf is scaled at 200 percent.

5. In the *Rotate* DRO, type the rotation angle in degrees.

The rotation angle is applied around the Z-axis of the drawing's origin.

6. Select the cutter compensation to be applied to the tool path: Click one of the following radio buttons:
 - *On*: the tool moves along the path.
 - *Outside / Right*: offsets the tool path right of the drawing path, seen from the direction where the tool enters the path.
 - *Inside / Left*: is the opposite of *Outside / Right*.

6.8.8.1 Working with Layers and Shapes

The .dxf file contains shapes grouped into layers.

In the *Shape Selection* tree view window, you can enable or disable individual layers and complete layers. You can select shapes either from the tree view window or in the *Preview* window.

PATHPILOT INTERFACE

To Change the Layer or Shape Cut Order

To change the shape order, use the *Up Arrow* and *Down Arrow* buttons above the shape selection tree view window. Shapes or layers higher in the tree view window are cut earlier than those below it.

The order in which the shapes are cut matches the order of the enabled shapes in the tree view window and the cyan path in the *Preview* window.

If a layer is selected, the whole layer is moved up or down. Shapes can't be moved between layers.

To Adjust the Tree View Window

Use the *Fold* and *Unfold* buttons to collapse and expand the layer and shape tree in the tree view window.

6.8.8.2 Working in the Preview Window

The *Preview* window uses the following colors:

- Selected paths are **cyan**
- Disabled paths are **gray**
- The drawing path is **white**
- The cut path is **magenta**
- The tool path between cuts is a **dark cyan stippled line**
- The coordinates are identified as follows:
 - The X-axis is **red**
 - The Y-axis is **green**
 - The Z-axis is **blue**

6.9 Probe Tab

The *Probe* tab contains automated functions to find X/Y/Z locations, set work offsets, probe pockets, slots or bosses, as well as instructions on probe and toolsetter setup and calibration. The *Probe* tab contains a separate, smaller notebook that consists of three tabs:

- X/Y/Z Probe
- Rect/Circ
- Probe/ETS Setup

PATHPilot INTERFACE

Tool 99 (the probe tool) must be the current tool in the spindle before using any of the probing functions. All probing moves occur at a feed rate specified by the current *F* command. This can be viewed and modified in the feed rate DRO. For more information, refer to *Feed Rate DRO* section earlier in this chapter.

NOTE: Check that probe polarity is set correctly on Settings screen before using the probing buttons by pressing probe tip while looking at probe indicator LED on Probe screen.

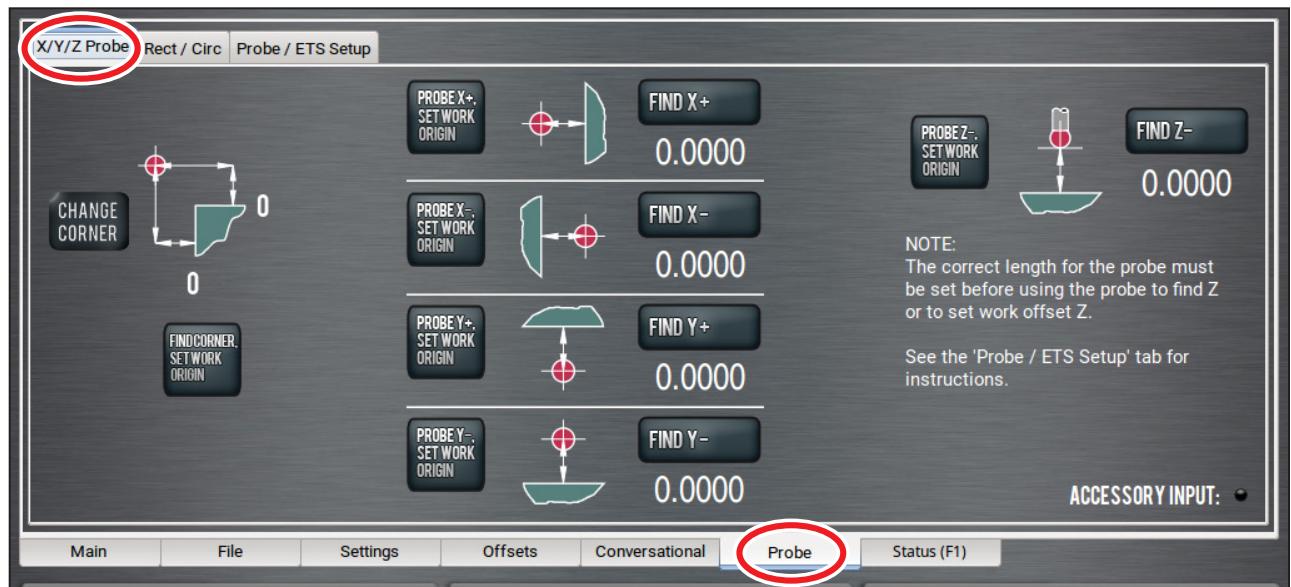


Figure 6.38

6.9.1 XYZ Probe Tab

The *X/Y/Z Probe* tab allows the user to quickly touch off a workpiece or vise jaw to find that feature's location in current work offset coordinates, or to touch off that feature and set the work offset zero to the feature's surface (see **Figure 6.38**).

The *Find Corner, Set Work Origin* button is used to probe the corner of a vise jaw or rectangular workpiece and set that corner to X/Y zero.

NOTE: To change the corner on which to probe a part, click *Change Corner*.

To use this button, first position the probe below the surface of the vise jaw and roughly 1" away from the vice jaw corner in X and Y (see **Figure 6.38**). Upon completion of the probing moves, the current active work offset system (e.g. G54) is set such that the vise jaw corner is 0, 0 (the X/Y origin).

The *Probe [Axis], Set Work Offset* buttons will probe in one axis only and set the current work offset origin to the probed surface for that axis. The direction of probing is specified by the + or - sign on the button and is described by the accompanying graphic.

PATHPILOT INTERFACE

The *Find [Axis]* buttons cause a probing move similar to the *Probe [Axis]*, *Set Work Offset* buttons, but will not change the work offset value. Instead, the location of the probed surface is displayed in the label below the button.

6.9.2 Rect/Circ Tab

The *Rect/Circ* tab contains buttons that automate tasks like finding the center of a pocket, slot, or bore, as well as finding the center of a circular or rectangular boss (see **Figure 6.39**). As the on-screen instructions suggest, first jog the probe below the top surface of the feature to be probed.

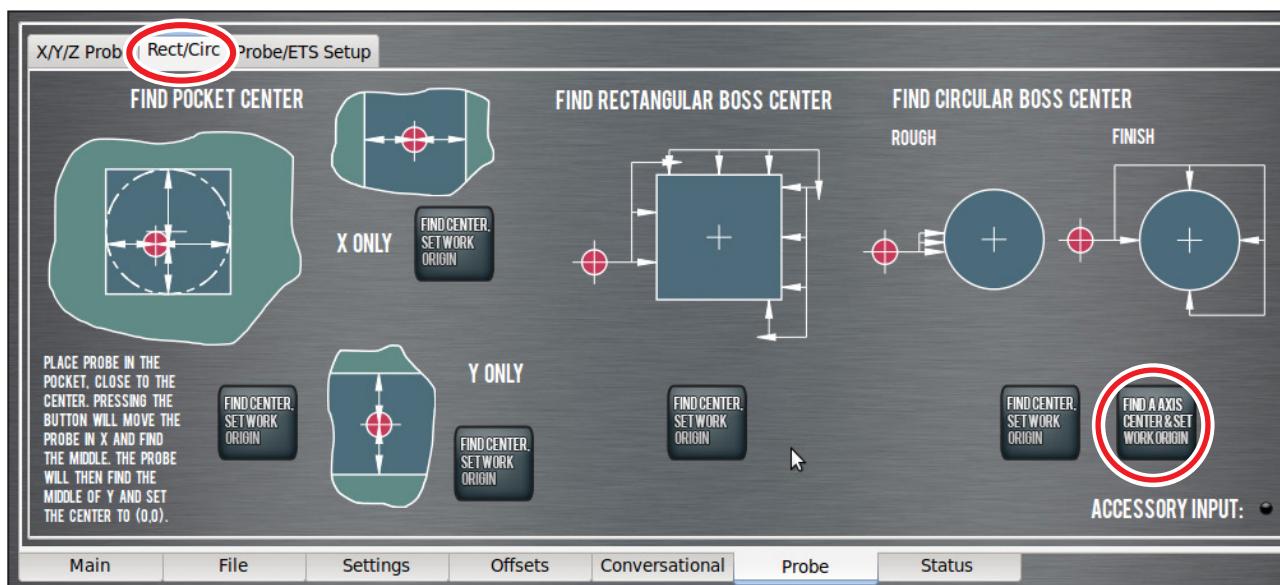


Figure 6.39

The *Find Pocket Center, Set Work Origin* button works in either a round or rectangular pocket. The probe moves in both X and Y to find the pocket center. The *Find Center, Set Work Origin* buttons perform a similar probing operation, but in X or Y only, and are intended to be used to find the center of a slot.

The rectangular boss center finding routine hunts around the edge of a square or rectangular workpiece to find the center. To use this routine, start with the probe below the top surface of the boss and on the left-hand side. Similarly, the circular boss center finding routine probes three times to find an approximate center of curvature, then confirms the circular boss center with four additional moves. To use this feature, start with the probe below the top surface of the boss and on the left-hand side.

The *Find A Axis Center & Set Work Origin* button is available for use with a 4th axis mounted in the A-axis orientation. The function probes a round workpiece mounted in the A-axis to find the center rotation of the A-axis. Move the probe to a point approximately directly above the A-axis center of rotation, and click the *Find A Axis Center & Set Work Origin* button.

PATHPilot INTERFACE

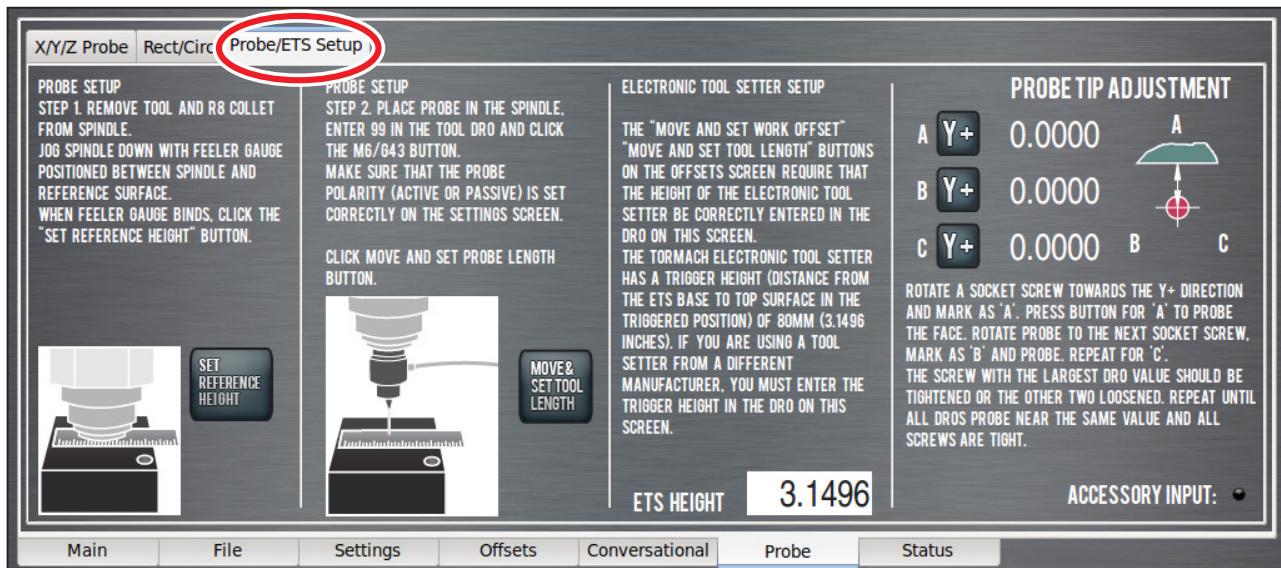


Figure 6.40

6.9.3 Probe/ETS Setup Tab

The *Probe/ETS Setup* tab is used to align and set the probe and ETS heights (see **Figure 6.40**). For information on probe and ETS setup, refer to chapter 8, *Accessories*.

6.10 ADMIN Commands

Several ADMIN commands are provided for operator use. Below is a list of commands used in the PathPilot interface:

ADMIN Command	Description
ADMIN CONFIG	Switch configuration
ADMIN DATE	Customize controller date and time
ADMIN DISPLAY	Customize controller screen display
ADMIN KEYBOARD	Customize controller keyboard layout
ADMIN NETWORK	WIFI network setup
ADMIN OPENDOORMAXRPM	Set spindle speed RPM with spindle door open; for use with Full Enclosure Door Switch Kit (PN 35550)
ADMIN SETTINGS BACKUP	Save tool offset and fixture information backup to store externally
ADMIN SETTINGS RESTORE	Restore tool offset and fixture information backup from external location
ADMIN TOUCHSCREEN	Calibrate touch screen; for use with 17" Touch Screen Kit (PN 35575)

7. PROGRAMMING

This chapter defines the languages (G-codes, etc.) that are understood and interpreted by the PathPilot® operating system, and is intended for reference purposes. If you want to learn about the principles of the control language so you can write programs by hand from first principles, consult an introductory textbook on G-code programming.

7.1 Definitions

The following terms are defined as follows:

PathPilot

This is the Tormach motion controller.

PathPilot Operating System (OS)

This is the PathPilot controller operating system.

Coordinate System

A coordinate system identifies the position of geometric features like points, lines, etc., in space. The default coordinate system in PathPilot is a standard right-hand coordinate system. This coordinate system is also known as a Cartesian coordinate system.

Linear Axes

The X-, Y- and Z-axes are the orthogonal lines that define a Cartesian Coordinate System. Position is measured in the active unit length specified by G20 (inches) or G21 (millimeters).

Origin

An origin is the location in a coordinate system where the position of each axis is equal to zero (X0 Y0 Z0). Each coordinate system can have only one origin.

Active Plane

There is always an active plane, which must be the XY-plane, the YZ-plane or the XZ-plane of the machining system. The Z-axis is perpendicular to the XY-plane, the X-axis to the YZ-plane and the Y-axis to the XZ-plane. Changing the active plane changes the interpretation of certain G-codes.

Units

The length units used to describe a position along the X-, Y- and Z-axes may be measured in either inches (G20 mode) or millimeters (G21 mode). Units for all other quantities involved in mill control cannot be changed. Different quantities use different specific units. Spindle speed is measured in revolutions per minute; rotational axes positions are measured in degrees; feed rates are expressed in current length units per minute, or in degrees per minute, as described above.

Rotational Axis

The A-axis is a rotational axis. In general, the axis of rotation can be collinear to a primary linear axis, or arbitrary. In usual practice, the axis of rotation of the A-axis is typically collinear to the X-axis. Position is measured in degrees. It is treated as a wrapped linear axis, meaning that the angular position increases without limit (goes toward plus infinity) as the axis turns counterclockwise and decreases without limit (goes toward minus infinity) as the axis turns clockwise. The direction of positive rotation is counterclockwise when viewed from the positive end of the corresponding X-, Y- or Z-axis.

Controlled Point

The controlled point is the point whose position and rate of motion are controlled. In practical application, this point is located somewhere along the spindle axis (Z-axis). The location of the controlled point can be moved out along the spindle axis by specifying some positive value for the tool length offset. This value is normally the length of the cutting tool in use, so that the controlled point is effectively located at the bottom center of the cutting tool.

Work Envelope

The work envelope is defined by the space that can be reached by the controlled point.

Current Position

The controlled point is always at a location called the current position and the operating system always knows where that is. Moving the controlled point changes the current position. The current position is defined by the values displayed on the digital readouts (DRO).

The current position can also be changed without any actual movement of the controlled point if any of several events take place:

- Length unit mode (G20/G21) is changed
- Tool length offset is changed
- Work offset is changed

Each of these events can change the values displayed in the DROs.

Coordinated Linear Motion

Coordinated linear motion describes a situation in which, nominally, each linear axis (X-, Y-, or Z-axis) moves at a constant speed and all axes move from their starting positions to their end positions at the same time. This produces motion in a straight line. Coordinated linear motion can be performed either at the prevailing feed rate or at rapid traverse rate. If physical limits on axis speed make the desired rate unobtainable, all axes are slowed to maintain the desired path.

In actual motion, it is often not possible to maintain constant speed because acceleration or deceleration is required at the beginning and/or end of the motion.

PROGRAMMING

It is possible, however, to control the axes so that, at all times, each axis has completed the same fraction of its required motion (as the other axes) and the tool maintains a straight line motion.

Arc and Helical Motion

Any pair of the linear axes (XY, YZ, and XZ) can be controlled to move in a circular arc in the plane of that pair of axes. While this is occurring, the third linear axis and/or the rotational axes can be controlled to move simultaneously at a constant rate. As in coordinated linear motion, these motions can be coordinated so that acceleration and deceleration do not affect the path. If the third linear axis moves simultaneously with arc motion, the trajectory of the controlled point forms a helix.

Feed Rate

The feed rate is the nominally steady rate at which the controlled point moves. Feed rates are programmed by the operator. The interpretation of the feed rate is detailed in the table below.

Motion	Feed Rate
Coordinated linear motion of one or more axis (X-, Y-, or Z-axis)	Inches per minute (G20 mode) or millimeters per minute (G21 mode)
Rotational axis motion of one axis (A-axis)	Degrees per minute
Coordinated linear motion of one or more axis (X-, Y-, or Z-axis) with simultaneous rotational axis motion (A-axis)	This type of motion is usually programmed in inverse time feed rate mode (G93)

Dwell

Commanding a dwell pauses the motion of the axes for a specific amount of time. The units in which you specify dwell are seconds; a decimal value is used to get less than one second.

Work Offsets

Work offsets allow you assign an origin to any location within the work envelope.

Up to nine different work offsets can be saved in the mill memory, but only one can be active at any given time. The default work offset is G54. The position of each work offset origin is stored in the Work Offset Table.

Tool Number

The tool number is used to identify a tool in a program. Each tool used in the program must have a unique tool number between 1 and 256.

Tool Table

The tool table stores the tool diameter value and tool length offset value associated with each tool number. The diameter value is used for cutter radius compensation. The tool length offset value is used to adjust the position of the controlled point for differences in the lengths among tools.

Feed and Speed Override Controls

The operating system has commands which enable (M48) or disable (M49) the feed and speed override slider controls. It is useful to be able to override these for some machining operations. Default settings in the program are set and the operator should not change them.

7.2 G-code Programming Language

7.2.1 Overview

The programming language of the mill is known as G-code. A G-code program is composed of one or more lines of code. Each line (called a block) may include commands to the machining system to do several different things. Blocks may be collected in a file to make a program.

A typical block consists of an optional line number at the beginning followed by one or more words. A word consists of a letter followed by a number (or strictly speaking, something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, G01 X3 is a valid line of code with two words. G01 is a command meaning move in a straight line at the programmed feed rate, and X3 provides an argument value (the value of X should be 3 at the end of the move). Most commands start with either G (general) or M (miscellaneous). The words for these commands are called G-codes and M-codes.

The language has two commands (M02 or M30), execution of either of which ends a program. A program may end before the end of a file. Lines of a file that occur after the end of a program are not to be executed in the normal flow, so generally they're parts of subroutines.

7.2.2 Block

A block (or equivalently line) of code is a section of programming language elements that are grouped together into a single statement. A program consists of one or more blocks, each separated by a line break. Blocks in a program are executed sequentially from top to bottom or until an end command (M02 or M30) is encountered.

7.2.3 Real Value

A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression or a unary operation value. Definitions of these follow in the *Word Initial Letters* table.

7.2.4 Number

Numbers are a subset of real values. Processing a real value to come up with a number is called evaluating. An explicit number evaluates to itself.

The following rules are used for explicit numbers. In these rules a digit is a single character between 0 and 9.

- A number consists of the following, in order: (1) an optional plus or minus sign, followed by (2) zero to many digits, followed, possibly, by (3) one decimal point, followed by (4) zero to many digits.
- There must be at least one digit somewhere in the number.
- There are two kinds of numbers: integers and decimals. An integer does not have a decimal point in it; a decimal does.
- Numbers may have any number of digits, subject to line length limitations. PathPilot only retains 17 significant figures. This is enough for all known applications.
- A non-zero number with no sign as the first character is assumed to be positive.

Initial zeros (before the decimal point and the first non-zero digit) and trailing zeros (after the decimal point and the last non-zero digit) are allowed but not required. A number written with initial or trailing zeros has the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes by the operating system are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close enough to an integer to be accepted as input. A decimal number which is supposed to be close to an integer is considered close enough if it is within 0.0001 of an integer.

7.2.5 Formatting G-code Blocks

A permissible block of input code consists of the following programming elements, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line:

- Optional block delete character (/)
- Optional line number
- Any number of words, parameter settings, and comments
- End of line marker (carriage return or line break)

Any input not explicitly allowed is illegal and causes the interpreter to signal an error or to ignore the line.

Programs are limited to 999,999 lines of code.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. For example, the line:

G00 x +0. 12 34y 7

is equivalent to:

G00 x+0.1234 y7

Blank lines are allowed in the input, but are ignored.

Input is not case sensitive, except in comments, therefore any letter outside a comment may be in uppercase or lowercase without changing the meaning of a line.

Block Delete Character

The operating system omits blocks of code that are prefixed with the forward slash symbol (/).

Line Number

A line number is indicated by the letter N followed by an integer (with no sign) between 0 and 99,999,999 and written without commas.

Line numbers may be repeated or used out of order, although normal practice is to avoid such usage. A line number is not required and often omitted.

Word

A word is a letter other than N or O followed by a real value. Words may begin with any of the letters shown in the table below. The table includes N and O for completeness, even though, as defined above, line numbers are not words. Several letters (I, J, K, L, P and R) may have different meanings in different contexts.

Word Initial Letters

Letter	Meaning
A	A-axis of mill
B	B-axis of mill
C	C-axis of mill
D	Tool radius compensation number
F	Feed rate
G	General function
H	Tool length offset index
I	X-axis offset for arcs X offset in G87 canned cycle
J	Y-axis offset for arcs Y offset in G87 canned cycle
K	Z-axis offset for arcs Z offset in G87 canned cycle
L	Number of repetitions in canned cycles/subroutines Key used with G10
M	Miscellaneous function
N	Line number
O	Subroutine label number

PROGRAMMING

Letter	Meaning
P	Dwell time in canned cycles Dwell time with G04 Key used with G10 Tapping depth in M871 – M874
Q	Feed increment in G83 canned cycle Repetitions of subroutine call
R	Arc radius Canned cycle retract level
S	Spindle speed
T	Tool selection
U	Synonymous with A
V	Synonymous with B
W	Synonymous with C
X	X-axis of mill
Y	Y-axis of mill
Z	Z-axis of mill

Parameter

Parameter programming is a special subset of the part programming language. For more details on the use of parameters, see *Advanced Programming with Parameters and Expressions* later in this chapter.

Comments and Messages

Comments can be added to lines of G-code to help clear up the intention of the programmer. Comments can be embedded in a line using parentheses () or for the remainder of a line using a semicolon. The semicolon is not treated as the start of a comment when enclosed in parentheses.

Comments may appear between words, but not between words and their corresponding parameter. So:

```
S100 (set speed) F200 (feed) is OK while  
S(speed) 100F(feed) is not.
```

If the comment occurs on a line with M00 or M01 and contains a file name with a .jpg or .png extension, PathPilot displays the image in the tool path window when it reaches a programmed M00 or M01 break.

1. Move an image file with a .jpg or .png extension to your PathPilot controller in one of the following locations:
 - In the same folder as the G-code program file
 - In a folder called images within the G-code program file's folder
 - In a folder called images within the home directory
2. Program an M00 or M01 break.
3. Using parentheses (), embed a comment within the line of G-code.
4. Type the file name of the image within the comment.

NOTE: Ensure the file name has either a .jpg or .png extension.

Example

M01 (photo_of_my_setup.jpg)

Display a Message - (MSG)

(MSG...) - displays a message if MSG appears after the left parenthesis and before any other printing characters. Variants of MSG which include white space and lowercase characters are allowed. The rest of the characters before the right parenthesis are considered to be a message. Messages are displayed on the *Status* screen.

Example:

(MSG, your message here) prints *your message here* to the *Status* screen.

7.2.6 Optional Program Stop Control – (M01 BREAK)

The optional program stop control (M01 BREAK) works as follows. If M01 break is on (indicated by the LED on the *M01 Break* button on the PathPilot interface) and a line in the G-code program contains an M01-code, program execution is stopped at the M01 line. The program resumes from that line when the *Cycle Start* button is pushed.

7.2.7 Additional G-code Formatting Notes

7.2.7.1 Repeated Items

A line may have any number of G words, but two G words from the same modal group may not appear on the same line. For more information, see *Modal Groups* later in this chapter.

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, #3=15 #3=6, for example, only the last setting takes effect. It is illogical but not illegal to set the same parameter twice on the same line.

7.2.7.2 Order of Execution

The order of items on a line does not determine the order of execution on the commands. For more information, see table *Order of Execution* later in this chapter.

The three types of items whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of items are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line which is as defined above.

If the second group (the parameter settings) is reordered, there is no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter takes effect. For example, after the line:

#3=15 #3=6

has been interpreted, the value of parameter 3 is 6. If the order is reversed to

#3=6 #3=15

and the line is interpreted, the value of parameter 3 is 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment is used. If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line.

For example, the line:

G40 G01 #3=15 (foo) #4=-7.0

has five items and means exactly the same thing in any of the 120 possible orders, such as #4=-7.0 G01 #3=15 g40 (foo), for the five items. The order of execution of items on a line is critical to safe and effective mill operation. If items occur on the same line, they are executed in a particular order. For more information, see *Order of Execution* table later in this chapter.

To impose a different order (e.g. to turn coolant off before the spindle is stopped), code the commands on separate blocks.

Order of Execution

Order	Item
1	Comment (including message)
2	Set feed rate mode (G93, G94, G95)
3	Set feed rate (F)
4	Set spindle speed (S)
5	Special I/O (M62 to M68) – currently not supported
6	Change tool (T)
7	Spindle on/off (M03, M05)
8	Save State (M70, M73, restore state (M72), invalidate state (M71)
9	Coolant on/off (M07, M08, M09)
10	Enable/disable overrides (M48, M49, M50, M51, M52, M53)
11	Operator defined commands (M100 to M199)
12	Dwell (G04)
13	Set active plane (G17, G18, G19)
14	Set length units (G20, G21)
15	Cutter radius compensation on/off (G40, G41, G42)
16	Tool table offset on/off (G43, G49)
17	Fixture table select (G54 – G58 and G59 P~)
19	Set path control mode (G61, G61.1, G64)
19	Set distance mode (G90, G91)
20	Set canned cycle return level mode (G98, G99)
21	Home, change coordinate system data (G10) or set offsets (G92, G94)
22	Perform motion (G00 to G03, G12, G13, G80 to G89 as modified by G53)
23	Stop (M00, M01, M02, M30, M60)

7.2.7.3 Error Handling

This section describes error handling in PathPilot. This operating system sometimes ignores things it does not understand. If a command does not work as expected or does nothing, check if it was typed correctly. The operating system does not check for excessively high-machining feeds or speeds. Nor does it detect situations where a legal command does something unfortunate, such as machining a fixture.

7.2.7.4 Modality and Modal Commands

G-codes and M-codes are, generally speaking, modal. Modal commands cause the machining system to change from one mode to another. The mode stays active until another command changes it implicitly or explicitly. For example, if coolant is turned on (M07 or M08) it stays on until it is explicitly turned off in the program (M09). A few G-codes and M-codes are non-modal. These codes have effect only on the lines on which they occur. For example, dwell (G04) is non-modal.

7.2.7.5 Modal Groups

Modal commands are arranged in sets called modal groups, and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time – for example inch units (G20) vs. millimeter units (G21). A machining system may be in many modes at the same time, with one mode from each modal group being in effect.

Modal Groups for G-codes

Group 1 =	{G00, G01, G02, G03, G33, G38.x, G73, G76, G80, G81, G82, G85, G86, G87, G88, G89} motion
Group 2 =	{G17, G18, G19, G17.1, G17.2, G17.3} plane selection
Group 3 =	{G90, G91} distance mode
Group 4 =	{G90.1, G91.1} arc IJK distance mode
Group 5 =	{G93, G94} feed rate mode
Group 6 =	{G20, G21} units
Group 7 =	{G40, G41, G42, G41.1, G42.1} cutter radius compensation
Group 8 =	{G43, G43.1, G49} tool length offset
Group 10 =	{G98, G99} return mode in canned cycles
Group 12 =	{G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3} coordinate system selection
Group 13 =	{G61, G61.1, G64} path control mode
Group 14 =	{G96, G97} spindle speed mode
Group 15 =	{G07, G08} lathe diameter mode

Modal Groups for M-codes

Group 4 =	{M00, M01, M02, M30, M60} stopping
Group 7 =	{M03, M05} spindle turning
Group 8 =	{M07, M08, M09} coolant (special case: M07 and M08 may be active at the same time)
Group 9 =	{M48, M49} enable/disable feed and speed override controls
Group 10 =	{operator defined M100 to M199}

Non-modal G-codes

Group 0 =	{G04, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}
-----------	---

7.2.7.6 Default Modes

For all G-code modal groups, when a machining system is ready to accept commands, one member of the modal group must be in effect. There are default settings for these modal groups. When the machining system is turned on or re-initialized, default values are automatically in effect.

Group 1, the first group on the table is a group of G-codes for motion. One of these is always in effect. That one is called the current motion mode.

7.3 G-codes

The supported G-codes are shown and described in more detail in this section. The descriptions contain command examples set in *Courier* type font.

Summary of G-codes

G00	Rapid positioning
G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Counter-clockwise circular interpolation
G04	Dwell
G07, G08	Diameter/radius mode – Do not use G08
G10 L1	Set tool table entry
G10 L10	Set tool table – calculated – workpiece
G10L11	Set tool table – calculated – fixture
G10 L2	Set work offset origin
G10 L20	Set work offset origin – calculated
G17, G18, G19	G19 only for lathe
G20/G21	Inch/millimeter unit
G28	Return home

(continued on next page...)

PROGRAMMING

Summary of G-codes (...continued)

G28.1	Reference axes
G30	Return home
G33	Spindle sync. motion (e.g. threading)
G33.1	Rigid tapping
G40	Cancel cutter radius compensation
G41/G42	Start cutter radius compensation left/right
G41.1, G42.1	<i>Dynamic Cutter Compensation</i>
G43	Apply tool length offset
G49	Cancel tool length offset
G53	Move in absolute machine coordinate system
G54	Use fixture offset 1
G55	Use fixture offset 2
G56-58	Use fixture offset 3, 4, 5
G59	Use fixture offset 6 / use general fixture number
G61/G61.1	Path control mode
G64	Path control with optional tolerance
G73	Canned cycle – peck drilling
G76	Multi-pass threading cycle
G80	Cancel motion mode (including canned cycles)
G81	Canned cycle – drilling
G82	Canned cycle – drilling with dwell
G83	Canned cycle – peck drilling
G85	Canned cycle – boring, no dwell, feed out
G86	Canned cycle – boring, spindle stop, rapid out
G88	Canned cycle – boring, spindle stop, manual out
G89	Canned cycle – boring, dwell, feed out
G90, G90.1	Absolute distance mode
G91, G91.1	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92 etc.
G93, G94, G95	Feed modes
G96, G97	CSS, RPM modes
G98	Initial level return / R-point level after canned cycles

In the command examples, the tilde symbol (~) stands for a real value. If L~ is written in an example, the ~ is often referred to as the L number. Similarly the ~ in H~ may be called the H number, and so on for any other letter. As described in detail elsewhere, a real value may be one of the following:

- An explicit number. For example: 4 . 4
- An expression. For example: [2+2 . 4]
- A parameter value, For example: #88
- A unary function value. For example: acos [0]

Many commands require axis words (X~, Y~, Z~, or A~) as an argument. Unless explicitly stated otherwise, the following assumptions can be made:

- Axis words specify a destination point
- Axis words relate to the currently active coordinate system, unless explicitly described as being in the absolute coordinate system
- Where axis words are optional, any omitted axes retain their current value

Any items in the command examples not explicitly described as optional are required.

7.3.1 Rapid Linear Motion – G00

For rapid linear motion, program: G00 X~ Y~ Z~ A~

Word	Definition
X~	X-axis coordinate
Y~	Y-axis coordinate
Z~	Z-axis coordinate
A~	A-axis coordinate

This produces coordinated linear motion to the destination point at the current traverse rate (or slower if the mill does not go that fast). It is expected that cutting won't take place when a G00 command is executing. It is an error if all axis words are omitted. The axis words are optional, except that at least one must be used. The G00 is optional if the current motion mode is G00.

If cutter radius compensation is active, the motion differs from the above; see *Cutter Compensation* later in this chapter. If G53 is programmed on the same line, the motion also differs; see *Absolute Coordinates* later in this chapter. Depending on where the tool is located, there are two basic rules to follow: If the Z value represents a cutting move in the positive direction (i.e. out of a hole), the X axis should be moved last. If the Z value represents a move in the negative direction, the X-axis should be executed first. It is an error if:

- All axis words are omitted
- G10, G28, G30 or G92 appear in the same block

PROGRAMMING

7.3.2 Linear Motion at Feed Rate – G01

For linear motion at feed rate (for cutting or not), program: G01 X~ Y~ Z~ A~ F~

Word	Definition
X~	X-axis coordinate
Y~	Y-axis coordinate
Z~	Z-axis coordinate
A~	A-axis coordinate
F~	Feed rate

This produces coordinated linear motion to the destination point at the current feed rate (or slower if the mill won't go that fast).

The axis words are optional, except that at least one must be used. The G01 is optional if the current motion mode is G01. If cutter radius compensation is active, the motion differs from the above; see *Cutter Compensation* later in this chapter. If G53 is programmed on the same line, the motion also differs; see *Absolute Coordinates* later in this chapter.

It is an error if:

- All axis words are omitted
- G10, G28, G30 or G92 appear in the same block
- No F word is specified

7.3.3 Arc at Feed Rate – G02, G03

A circular or helical arc is specified using either G02 (clockwise arc) or G03 (counterclockwise arc) as shown in **Figure 7.1** and **Figure 7.2**. The axis of the circle or helix must be parallel to the X-, Y- or Z-axis of the mill coordinate system. The axis (or equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane) or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

If a line of code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. Lines of this sort are hardly ever programmed.

If cutter radius compensation is active, the motion will differ from the above; see *Cutter Compensation* later in this chapter.

Two formats are allowed for specifying an arc: the center format and the radius format. In both formats the G02 or G03 is optional if it is the current motion mode.

7.3.3.1 Radius Format Arc

For an arc in radius format, program: G02 X~ Y~ Z~ A~ R~ (for a clockwise arc) or G03 X~ Y~ Z~ A~ R~ (for a counterclockwise arc).

Word	Definition
X~	X-axis coordinate
Y~	Y-axis coordinate
Z~	Z-axis coordinate
A~	A-axis coordinate
R~	Radius of arc

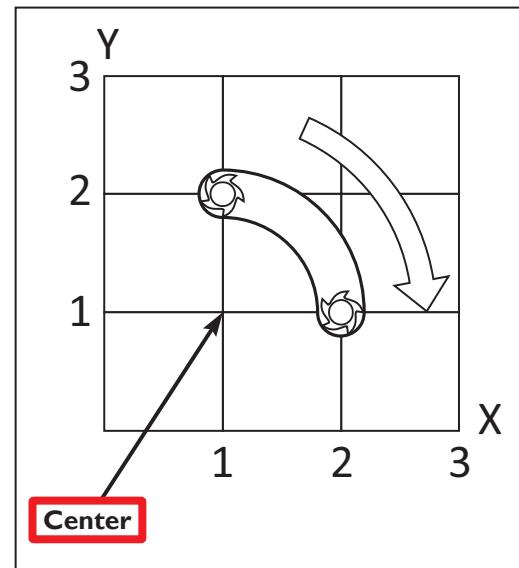


Figure 7.1

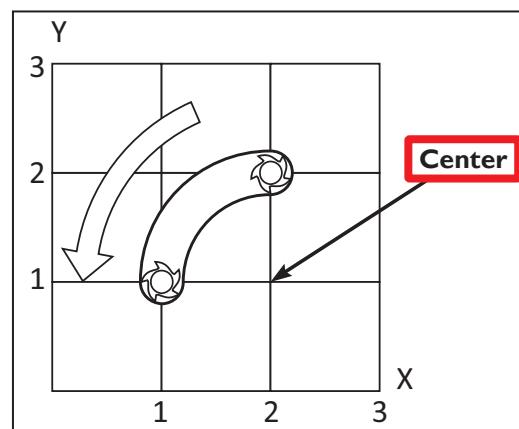


Figure 7.2

In radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. R is the radius. The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used. The R number is the radius. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees.

If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified.

It is an error if:

- Both of the axis words for the axes of the selected plane are omitted
- No R word is given
- End point of the arc is the same as the current point
- G10, G28, G30 or G92 appear in the same block

It is not good practice to program radius format arcs that are nearly full circles or are semicircles (or nearly semicircles) because a small change in the location of the end point produces a much larger change in the location of the center of the circle (and, hence, the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts. Nearly full circles are outrageously bad, semicircles (and nearly so) are only very bad. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK.

Here is an example of a radius format command to mill an arc:

```
G17 G02 X 1.0 Y 1.5 R 2.0 Z 0.5
```

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=1.0, Y=1.5 and Z=0.5, with a radius of 2.0. If the starting value of Z is 0.5, this is an arc of a circle parallel to the XY-plane; otherwise it is a helical arc.

7.3.3.2 Center Format Arc

For an arc in center format, program: G02 X~ Y~ Z~ I~ J~ (for a clockwise arc) or G03 X~ Y~ Z~ I~ J~ (for a counterclockwise arc).

Word	Definition
X~	X-axis coordinate
Y~	Y-axis coordinate
Z~	Z-axis coordinate
A~	A-axis coordinate

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point.

The center is specified using the two I, J, K words associated with the active plane. These specify the center relative to the current point at the start of the arc, defined in incremental coordinates from the start point.

It is an error if:

- When the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inches (if inches are being used) or 0.002 millimeters (if millimeters are being used)
- G10, G28, G30 or G92 appear in the same block

Arc in XY Plane

When the XY-plane is selected, program: G02 X~ Y~ Z~ A~ I~ J~ (or use G03 instead of G02). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current location or coordinates – depending on arc distance mode (G90.1/G91.1) of the center of the circle (X and Y directions, respectively). I and J are optional except that at least one of the two must be used.

It is an error if:

- X and Y are both omitted
- I and J are both omitted

Arc in XZ Plane

When the XZ-plane is selected, program: G02 X~ Y~ Z~ A~ I~ K~ (or use G03 instead of G02). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location or coordinates – depending on arc distance mode (G90.1/G91.1) of the center of the circle (X and Z directions, respectively). I and K are optional except that at least one of the two must be used.

It is an error if:

- X and Z are both omitted
- I and K are both omitted

Arc in YZ Plane

When the YZ-plane is selected, program: G02 X~ Y~ Z~ A~ J~ K~ (or use G03 instead of G02). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location or coordinates – depending on depending on arc distance mode (G90.1/G91.1) of the center of the circle (Y and Z directions, respectively). J and K are optional except that at least one of the two must be used.

It is an error if:

- Y and Z are both omitted
- J and K are both omitted

PROGRAMMING

Here is an example of a center format command to mill an arc in incremental arc distance mode (G91.1):

```
G17 G02 X1.0 Y1.6 I0.3 J0.4 Z0.9
```

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=1.0, Y=1.6 and Z=0.9, with its center offset in the X direction by 0.3 units from the current X location and offset in the Y direction by 0.4 units from the current Y location. If the current location has X=0.7, Y=0.7 at the outset, the center is at X=1.0, Y=1.1. If the starting value of Z is 0.9, this is a circular arc; otherwise it is a helical arc. The radius of this arc would be 0.5.

In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

```
(Sample Program G02EX3:  
(Workpiece Size: X4, Y3, Z1)  
(Tool: Tool #2, 1/4" Slot Drill)  
(Tool Start Position: X0, Y0, Z1)  
N2 G90 G80 G40 G54 G20 G17 G94 G64 (SAFETY BLOCK)  
N5 G90 G20  
N10 M06 T2 G43 H2  
N15 M03 S1200  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.1 F5  
N35 G02 X2 Y2 I1 J0 F20 (ARC FEED CW, RADIUS I1,J0 AT 20 IPM)  
N40 G01 X3.5  
N45 G02 X3 Y0.5 R2 (ARC FEED CW, RADIUS 2)  
N50 X1 Y1 R2 (ARC FEED CW, RADIUS 2)  
N55 G00 Z0.1  
N60 X2 Y1.5  
N65 G01 Z-0.25  
N70 G02 X2 Y1.5 I0.25 J-0.25 (FULL CIRCLE ARC FEED MOVE CW)  
N75 G00 Z1  
N80 X0 Y0  
N85 M05  
N90 M30
```

7.3.4 Dwell – G04

For a dwell, program: G04 P~

Word	Definition
P~	Dwell time (measured in seconds)

Dwell keeps the axes unmoving for the period of time in seconds specified by the P number.

Example: G04 P4.2 (to wait 4.2 seconds)

It is an error if:

- The P number is negative

7.3.5 Set Offsets – G10

Use the buttons and DROs on the *Offsets* screen to set offsets; they can be set programmatically via the G10 G-code command.

7.3.5.1 Set Tool Table – G10 L1

To define an entry in the tool table, program: G10 L1 P~ X~ Y~ R~ I~ J~ Q~

Word	Definition
P~	Tool number
R~	Radius of tool

G10 L1 sets the tool table for the P tool number to the values of the words.

A valid G10 L1 rewrites and reloads the tool table.

Example: G10 L1 P2 R0.015 Q3 (setting tool 2 radius to 0.015 and orientation to 3).

It is an error if:

- *Cutter Compensation* is on
- The P number is unspecified
- The P number is not a valid tool number from the tool table
- The P number is 0

7.3.5.2 Set Coordinate System – G10 L2

To define the origin of a work offset coordinate system, program: G10 L2 P-

Word	Definition
P~	Number of coordinate system to use (G54 = 1, G59.3 = 9)

PROGRAMMING

Important Concepts:

The G10 L2 PN command does not change from the current coordinate system to the one specified by P, use G54-59.3 to select a coordinate system.

If a G92 origin offset was in effect before G10 L2, it continues to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed. If it is currently active, the new coordinates take effect immediately.

It is an error if:

- The P number does not evaluate to an integer in the range 0 to 9
- An axis other than X or Z is programmed

7.3.5.3 Set Tool Table – G10 L10

To change the tool table entry for tool P so that if the tool offset is reloaded, with the mill in its current position and with the current G5x and G92 offsets active, program:

G10 L10 P~ Z~ R~ I~ J~ Q~

Word Definition

P~	Tool number
R~	Radius of tool

The current coordinates for the given axes become the given values. The axes that are not specified in the G10 L10 command are not changed. This could be useful with a probe move as described in the G38 section.

It is an error if:

- *Cutter Compensation* is on
- The P number is unspecified
- The P number is not a valid tool number from the tool table
- The P number is 0

7.3.5.4 Set Tool Table – G10 L11

G10 L11 is just like G10 L10 except that instead of setting the entry according to the current offsets, it is set so that the current coordinates would become the given value if the new tool offset is reloaded and the mill is placed in the G59.3 coordinate system without any G92 offset active. This allows the operator to set the G59.3 coordinate system according to a fixed point on the mill, and then use that fixture to measure tools without regard to other currently active offsets.

Program: G10 L11 P~ X~ Z~ R~ I~ J~ Q~

Word	Definition
P~	Tool number
R~	Radius of tool

It is an error if:

- *Cutter Compensation* is on
- The P number is unspecified
- The P number is not a valid tool number from the tool table
- The P number is 0

7.3.5.5 Set Coordinate System – G10 L20

G10 L20 is similar to G10 L2 except that instead of setting the offset/entry to the given value, it is set to a calculated value that makes the current coordinates become the given value.

Program: G10 L20 P~ X~ Y~ Z~ A~

Word	Definition
P~	Number of coordinate system to use (G54 = 1, G59.3 = 9)
X~	X-axis coordinate
Y~	Y-axis coordinate
Z~	Z-axis coordinate
A~	A-axis coordinate

It is an error if:

- The P number does not evaluate to an integer in the range 0 to 9
- An axis other than X or Z is programmed

7.3.6 Plane Selection – G17, G18, and G19

To select the XY-plane as active, program: G17 (see **Figure 7.3**).

To select the XZ-plane as active, program: G18 (see **Figure 7.4**)

To select the YZ-plane as active, program: G19

The active plane determines how the tool path of an arc (G02 or G03) or canned cycle (G73, G81-G89) is interpreted.

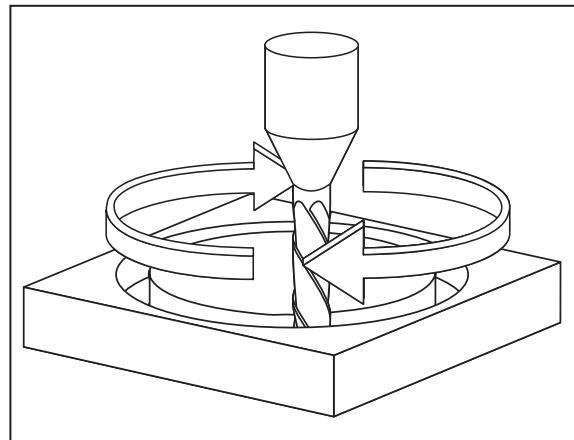


Figure 7.3

7.3.7 Length Units – G20, G21

To set length units to inches, program: G20

To set length units to millimeters, program: G21

It is best practice to program either G20 or G21 near the beginning of a program, before any motion occurs. Also, avoid using either one anywhere else in the program. It is the responsibility of the operator to make sure all numbers are appropriate for use with the current length units.

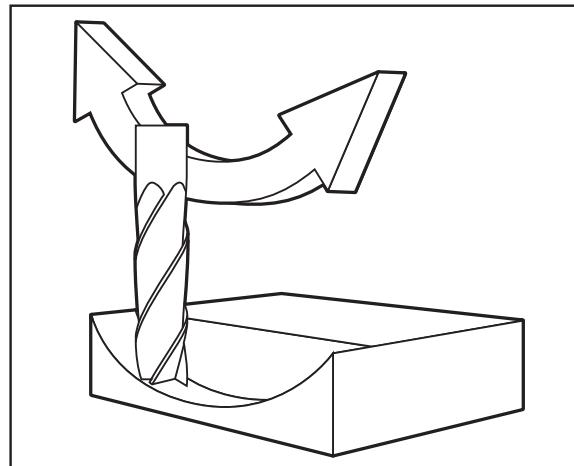


Figure 7.4

7.3.8 Return to Pre-defined Position – G28, G28.1

To make a rapid linear move to the G28.1 position, program: G28

To make a rapid linear move to the G28.1 position by first going to the intermediate position specified by the X~, Y~, and Z~ words, program: G28 X~Y~Z~

To store the current location of the tool in the G28.1 setting, program: G28 . 1

G28 uses the values stored in parameters 5161, 5162, and 5163 as the X,Y, and Z final points to move to. The parameter values are absolute mill coordinates in the native machine units of inches.

G28.1 stores the current absolute position into parameters 5161-5163.

It is an error if:

- *Cutter Compensation* is turned on

7.3.9 Return to Pre-defined Position – G30, G30.1

G30 uses the values stored in parameters 5181 and 5183 as the X and Z final point to move to. The parameter values are absolute mill coordinates in the native machine units of inches.

G30 makes a rapid traverse move from the current position to the absolute position of the values in parameters.

G30 X~ Z~ makes a rapid traverse move to the position specified by axes including any offsets, then makes a rapid traverse move to the absolute position of the values in parameters 5181 and/or 5183. Any axis not specified won't move.

G30.1 stores the current absolute position into parameters 5181-5183.

It is an error if:

- *Cutter Compensation* is turned on

7.3.10 Straight Probe – G38.x

G38.2 – probe toward workpiece, stop on contact, signal error if failure

G38.3 – probe toward workpiece, stop on contact

G38.4 – probe away from workpiece, stop on loss of contact, signal error if failure

G38.5 – probe away from workpiece, stop on loss of contact

G38.6 – move away from the workpiece ignoring probe input

To perform a straight probe operation program: G31 X~ Y~ Z~ A~

The probe will conventionally be tool #99. The rotational axis words are allowed, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The linear axis words are optional, except that at least one of them must be used. The tool in the spindle must be a probe. It is an error if:

- The current point is less than 0.01 inch (0.254 millimeter) from the programmed point;
- G38 is used in inverse time feed rate mode;
- Any rotational axis is commanded to move;
- No X-, Y- or Z-axis word is used.
- Feed rate is zero
- Probe is already tripped

In response to this command, the mill moves the controlled point (which should be at the end of the probe tip) in a straight line at the current feed rate toward the programmed point; if the probe trips, then the probe decelerates.

After successful probing, parameters 5061 to 5064 will be set to the coordinates of the location of the controlled point at the time the probe tripped (not where it stopped), or if it does not trip to the coordinates at the end of the move and a triplet giving X, Y and Z at the trip is written to the triplet file.

7.3.10.1 Using the Straight Probe Command

Using the straight probe command, if the probe shank is kept nominally parallel to the Z-axis (i.e., any rotational axes are at zero) and the tool length offset for the probe is used, so that the controlled point is at the end of the tip of the probe:

- Without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane may, for example, be found;
- If the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane may, for example, be found;
- If the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known approximately, the center of a circular hole, may, for example, be found;
- If the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, more uses may be made of the straight probe command, such as finding the diameter of a circular hole.

Example code:

```
o<probe_pocket> sub
(probe to find center of circular or rectangular pocket)
#<x_start> = #5420      (Current X Location)
#<y_start> = #5421      (Current Y Location)

#<x_max> = 1
#<x_min> = -1
#<y_max> = 1
#<y_min> = -1

#<feed_rate> = 30      (30 IPM)

F #<feed_rate>
G38.3 X #<x_max>      (rough probe +X side of hole)
F [#<feed_rate>/30]
G38.5 X #<x_start>    (finish probe)
#<x_plus>=#5061        (save results)
G00 X #<x_start>      (return to start)

F #<feed_rate>
G38.3 X #<x_min>      (probe -X side of hole)
F [#<feed_rate>/30]
```

```

G38.5 X #<x_start>
#<x_minus>=#5061                                (save results)
G00 X #<x_start>
#<x_center> = [ [#<x_plus>+#<x_minus>]/2 ]
G00 X #<x_center>                               (go to middle)

F #<feed_rate>
G38.3 Y #<y_max>      (probe +Y side of hole)
F [#<feed_rate>/30]
G38.5 Y #<y_start>
#<y_plus>=#5062        (save results)
G00 Y #<y_start>      (return to start)

F #<feed_rate>
G38.3 Y #<y_min>      (probe -Y side of hole)
F [#<feed_rate>/30]
G38.5 Y #<y_start>
#<y_minus>=#5062       (save results)
G00 Y #<y_start>
#<y_center> = [ [#<y_plus>+#<y_minus>]/2 ]
G00 Y #<y_center>     (go to middle)

G10 L20 P1 X 0 Y 0  (set current location to zero)

F #<feed_rate>  (restore original feed rate)

o<probe_pocket> endsub
M02

```

7.3.11 Cutter Compensation – G40, G41, and G42

7.3.11.1 Cutter Compensation Off – G40

To turn *Cutter Compensation* off, program: G40

It is OK to turn compensation off when it is already off.

It is an error if:

- A G02/G03 arc move is programmed next after a G40
- The linear move after turning compensation off is less than twice the tool tip radius

7.3.11.2 Cutter Compensation On – G41, G42

To program *Cutter Compensation* to the left of the programmed tool path, program: G41 D~

To program *Cutter Compensation* to the right of the programmed tool path, program: G42 D~

Word	Definition
D~	Tool number associated with the diameter offset to be applied

The D word is optional; if there is no D word the radius of the currently loaded tool is used (if no tool is loaded and no D word is given, a radius of 0 is used).

If supplied, the D word is the tool number to use.

To start *Cutter Compensation* to the left of the part profile, use G41. G41 starts *Cutter Compensation* to the left of the programmed line as viewed looking down on the mill.

To start *Cutter Compensation* to the right of the part profile, use G42. G42 starts *Cutter Compensation* to the right of the programmed line as viewed looking down on the mill.

The lead in move must be at least as long as the tool radius. The lead in move can be a rapid move. Operator M100-M199 commands are allowed when *Cutter Compensation* is on.

It is an error if:

- The D number is not a valid tool number or 0
- *Cutter Compensation* is commanded to turn on when it is already on

7.3.12 Dynamic Cutter Compensation – G41.1, G42.1

To program dynamic *Cutter Compensation* to the left of the programmed tool path, program: G41.1 D~

To program dynamic *Cutter Compensation* to the right of the programmed tool path, program: G42.1 D~

Word	Definition
D~	Tip radius multiplied by 2

G41.1 and G42.1 function the same as G41 and G42 with the added scope of being able to ignore the tool table and to program the tool diameter. The L word defaults to 0 if unspecified.

It is an error if:

- The L number is not in the range from 0 to 9 inclusive
- *Cutter Compensation* is commanded to turn on when it is already on

7.3.13 Apply Tool Length Offset – G43

To apply a tool length offset from a stored value in the tool table, program: G43 H~

Word	Definition
H~	Tool number associated with the length offset to be applied. Generally speaking, the value of the H~ Word should match the active tool number (T~ Word)

It is an error if:

- The H number is not an integer, or
- The H number is negative, or
- The H number is not a valid tool number

It is OK to program using the same offset already in use. It is also OK to program without a tool length offset if none is currently being used.

7.3.14 Engrave Sequential Serial Number – G47

To engrave a serial number, either alone or added to the end of any desired text, program: Z~ R~ X~ Y~ P~ Q~ D~

Word Definition

Z~	The depth of cut of the engraving.
R~	The retract height between character segments in the numbers.
X~	If present, specifies the starting 'X' position, or the left side of the serial number. If omitted, the current X position is assumed.
Y~	If present, specifies the starting 'Y' position, or the bottom side of the serial number. If omitted, the current Y position is assumed.
P~	If present, is the 'X' extent (width) in current units (inches or millimeters) of the engraved number.
Q~	If present, is the 'Y' extent (height) in current units (inches or millimeters) of the engraved number.
D~	If present, is the requested number of decimals of the engraved number. If the requested D value is more than the number of decimals in the serial number, the serial number will show leading zeros. If the requested D value is less than the number of decimals in the serial number, only the digits of the serial number will show. For example: a serial number of 10 where D = 4 engravés as 0010; a serial number of 9056 where D = 3 engravés as 9056.

It is an error if:

- Cutter Compensation is on
- The Z number is unspecified
- The R number is unspecified
- The Z number is greater than the R number
- The P number is too small (determined by the font used)
- The Q number is too small (determined by the font used)

7.3.15 G49 Cancel Tool Length Compensation – G49

To cancel tool length compensation, program: G49

7.3.16 Absolute Coordinates – G53

For rapid linear motion to a point expressed in absolute coordinates, program:

G01 G53 X~ Y~ Z~ (or similarly with G00 instead of G01), where all the axis words are optional, except that at least one must be used. The G00 or G01 is optional if it is in the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This produces coordinated linear motion to the programmed point. If G01 is active, the speed of motion is the current feed rate (or slower if the mill won't go that fast). If G00 is active, the speed of motion is the current traverse rate (or slower if the mill won't go that fast).

It is an error if:

- G53 is used without G00 or G01 being active
- G53 is used while cutter radius compensation is on

7.3.17 Select Work Offset Coordinate System – G54 to G59.3

To select a work offset coordinate system, program: G54, G55, etc, as defined in the table below.

GXX	Definition
G54	Select Coordinate System 1
G55	Select Coordinate System 2
G56	Select Coordinate System 3
G57	Select Coordinate System 4
G58	Select Coordinate System 5
G59	Select Coordinate System 6
G59.1	Select Coordinate System 7
G59.2	Select Coordinate System 8
G59.3	Select Coordinate System 9

It is an error if:

- One of these G-codes is used while cutter radius compensation is on

The X- and Z-axis work offset values are stored in parameters corresponding to the system in use (e.g. System 1 X=5221, Z=5223; System 2 X=5141, Z=5143; up to System 9 X= 5381, Z = 5383).

7.3.18 Set Exact Path Control Mode – G61

To put the machining system into exact path mode, program: G61

7.3.19 Set Blended Path Control Mode – G64

To attempt to maintain the defined feed velocity, program: G64 P~ Q~

Word	Definition
P~	If present, specifies the maximum acceptable tool path deviation to round corners to maintain speed. If P is omitted then the speed is maintained however far from the programmed path the tool cuts.
Q~	If present, specifies the maximum deviation from collinearity that will collapse a series of linear G01 moves at the same feed rate into a single linear move.

NOTE: It is OK to program for the mode that is already active.

7.3.20 Distance Mode – G90, G91

Interpretation of the operating system-code can be in one of two distance modes: absolute (see **Figure 7.5**) or incremental (see **Figure 7.6**).

To go into absolute distance mode, program: G90. In absolute distance mode, axis numbers (X, Y, Z, A) usually represent positions in terms of the currently active coordinate system. Any exceptions to that rule are described explicitly in this section.

To go into incremental distance mode, program: G91. In incremental distance mode, axis numbers (X, Y, Z, A) usually represent increments from the current values of the numbers. I and J numbers always represent increments, regardless of the distance mode setting. K numbers represent increments.

7.3.21 Arc Distance Mode – G90.1, G91.1

G90.1 – Absolute distance mode for I, and K offsets. When G90.1 is in effect I and K both must be specified with G02/3 for the XZ plane or it is an error.

G91.1 – Incremental distance mode for I, and K offsets. G91.1 Returns I and K to their default behavior.

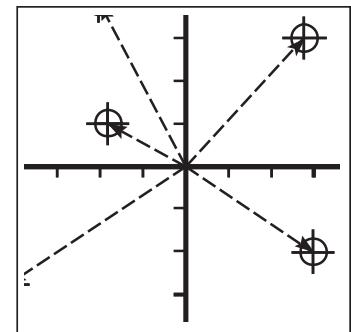


Figure 7.5

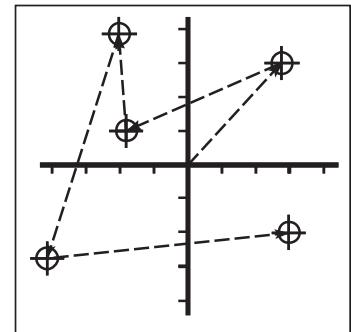


Figure 7.6

7.3.22 Temporary Work Offsets – G92, G92.1, G92.2, and G92.3

To apply a temporary work offset, program: G92 X~ Y~ Z~ A~

Word	Definition
X~	X-axis coordinate
Y~	Y-axis coordinate
Z~	Z-axis coordinate
A~	A-axis coordinate

This is a legacy feature. Most modern programming methods do not use temporary work offsets.

G92 reassigns the current controlled point to the coordinates specified by the axis words (X~, Y~, Z~, and/or A~). No motion takes place.

The axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. Incremental distance mode (G91) has no effect on the action of G92.

When G92 is executed, it is applied to the origins of all coordinate systems (G54-G59.3). For example, suppose the current controlled point is at X=4 and there is currently no G92 offset active. Then G92 X7 is programmed. This reassigns the current controlled point to X=7, effectively moving the origin of the active coordinate system -3 units in X. The origins of all inactive coordinate systems also move -3 units in X. This -3 is saved in parameter 5211.

G92 offsets may be already be in effect when the G92 is called. If this is the case, the offset is replaced with a new offset that makes the current point become the specified value. It is an error if:

- All axis words are omitted

The operating system stores the G92 offsets and reuses them on the next run of a program. To prevent this, one can program a G92.1 (to erase them), or program a G92.2 (to stop them being applied – they are still stored).

G92.1 – Reset axis offsets to zero and sets parameters 5211 - 5219 to zero

G92.2 – Reset axis offsets to zero

G92.3 – Sets the axis offset to the values saved in parameters 5211 to 5219

7.3.23 Feed Rate Mode – G93, G94, and G95

To set the active feed rate mode to inverse time, program: G93

Inverse time is used to program simultaneous coordinated linear and coordinated rotary motion. In inverse time feed rate mode, an F word means the move should be completed in [one divided by the F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute.

PROGRAMMING

When the inverse time feed rate mode is active, an F word must appear on every line which has a G01, G02, or G03 motion, and an F word on a line that does not have G01, G02, or G03 is ignored. Being in inverse time feed rate mode does not affect G00 (rapid traverse) motions.

To set the active feed rate mode to units per minute mode, program: G94

In units per minute feed rate mode, an F word is interpreted to mean the controlled point should move at a certain number of inches per minute, or millimeters per minute, depending upon what length units are being used.

To set the active feed rate mode to units per revolution mode, program: G95

In units per revolution mode, an F word is interpreted to mean the controlled point should move a certain number of inches per revolution of the spindle, depending on what length units are being used. G95 is not suitable for threading, for threading use G33 or G76.

It is an error if:

- Inverse time feed rate mode is active and a line with G01, G02, or G03 (explicitly or implicitly) does not have an F word
- A new feed rate is not specified after switching to G94 or G95 canned cycle return level – G98 and G99

7.3.24 Spindle Control Mode – G96, G97

To set constant surface speed mode, program: G96 D~ S~

Word Definition

D~	Maximum spindle RPM. This word is optional
S~	Surface speed. If G20 is active mode, the value is interpreted as feet per minute. If G21 is active mode, the value is interpreted as meters per minute

Example:

G96 D2500 S250 (set constant surface speed with a maximum rpm of 2500 and a surface speed of 250).

It is an error if:

- S is not specified with G96
- A feed move is specified in G96 mode while the spindle is not turning

When using G96 (the most common mode of mill operation), X0 in the current coordinate system (including offsets and tool lengths) must be the spindle axis.

To set RPM mode, program: G97

7.4 Canned Cycles

The canned cycles described in the table below are implemented in PathPilot.

Canned Cycle	Description
G80	Cancel active canned cycle
G81	Simple drilling cycle
G82	Simple drilling with dwell cycle
G83	Peck drilling cycle
G73	High speed peck drilling cycle
G84	Tapping cycle
G85	Boring cycle – feedrate out
G86	Boring cycle – stop, rapid out
G88	Boring cycle – stop, manual out
G89	Boring cycle – dwell, feedrate out

All canned cycles are performed with respect to the active plane. The descriptions in this section assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Word	Definition
X~	X-axis coordinate
Y~	Y-axis coordinate
Z~	Z-axis coordinate
A~	A-axis coordinate
R~	Retract position along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane)
L~	L number is optional and represents the number of repeats

All canned cycles use X, Y, Z, and R words. The R word sets the retract position; this is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

Rotational axis (A-axis) words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

The R number is always sticky. Sticky numbers keep their value on subsequent blocks if they are not explicitly programmed to be different.

In absolute distance mode (G90), the X, Y, R and Z numbers are absolute positions in the current coordinate system. In incremental distance mode (G91), when the XY-plane is selected, X, Y and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ- or XZ-plane is selected, treatment of the axis words is analogous.

Many canned cycles use the L word. The L word is optional and represents the number of repeats. L0 is not allowed. The L word is not sticky. The interpretation of the L word depends on the active distance mode:

- In incremental distance mode (G91), L > 1 in incremental mode means (with the XY-plane selected), that the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first iteration) or to the X and Y positions at the end of the previous go-around (on the subsequent repetitions). The R and Z positions do not change during the repeats
- In absolute distance mode (G90), L > 1 means do the same cycle in the same place several times. Omitting the L word is equivalent to specifying L=1

The height of the retract move at the end of each repeat (called clear Z in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98) or otherwise to the R position.

It is an error if:

- X, Y and Z words are all missing during a canned cycle
- A P number is required and a negative P number is used
- An L number is used that does not evaluate to a positive integer
- Rotational axis motion is used during a canned cycle
- Inverse time feed rate is active during a canned cycle
- Cutter radius compensation is active during a canned cycle

When the XY plane is active, the Z number is sticky and it is an error if:

- The Z number is missing and the same canned cycle was not already active
- The R number is less than the Z number

When the XZ plane is active, the Y number is sticky and it is an error if:

- The Y number is missing and the same canned cycle was not already active
- The R number is less than the Y number

When the YZ plane is active, the X number is sticky and it is an error if:

- The X number is missing and the same canned cycle was not already active
- The R number is less than the X number

At the very beginning of the execution of any of the canned cycles (with the XY-plane selected), if the current Z position is below the R position, the Z-axis will move in rapid motion to the R position. This happens only once, regardless of the value of L. In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

- A straight traverse parallel to the XY-plane to the given XY-position
- A straight traverse of the Z-axis only to the R position, if it is not already at the R position

If the XZ- or YZ-plane is active, the preliminary and in-between motions are analogous.

7.4.1 High Speed Peck Drilling Cycle – G73

The G73 cycle is intended for deep drilling with chip breaking (see **Figure 7.7**). The retracts in this cycle break the chip but do not totally retract the drill from the hole. It is suitable for tools with long flutes which clear the broken chips from the hole. This cycle takes a Q number which represents a delta increment along the Z-axis. Program: G73 X~ Z~ R~ L~ Q~

Word	Definition
Q~	Delta increment along Z axis

Step #	Description
1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
3	Rapid back incrementally in Z 0.010"
4	Repeat steps 1, 2 and 3 until the Z position is reached at step 1
5	Rapid back down to the current hole bottom, backed off a bit
6	Retract the Z-axis at traverse rate to clear Z

It is an error if:

- The Q number is negative or zero
- The R number is not specified

7.4.2 Cancel Active Canned Cycle – G80

The G80 cycle cancels all canned cycles. Program: G80

It is OK to program G80 if no canned cycles are in effect. After a G80, the motion mode must be set with G00 or any other motion mode G word. If motion mode is not set after G80, this error message appears: *Cannot use axis values without a g code that uses them.*

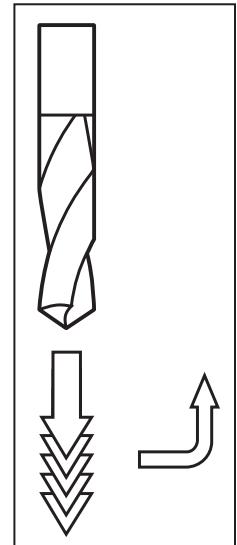


Figure 7.7

7.4.3 Simple Drilling Cycle – G81

The G81 cycle is intended for drilling. Program: G81 X~ Y~ Z~ A~ R~ L~

Step #	Description
1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate to the Z position
3	Retract the Z-axis at traverse rate to clear Z

The following examples demonstrate how the G81 canned cycle works in detail. Other canned cycles work in a similar manner.

Example 1: Suppose the current position is (1, 2, 3) and the XY-plane has been selected and the following line of NC-code is interpreted.

G90 G81 G98 X4 Y5 Z1.5 R2.8

This calls for absolute distance mode (G90), old “Z” retract mode (G98) and calls for the G81 drilling cycle to be performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. The following moves take place:

Step #	Description
1	G00 motion parallel to the XY-plane to (4,5,3)
2	G00 motion parallel to the Z-axis to (4,5,2.8)
3	G01 motion parallel to the Z-axis to (4,5,1.5)
4	G00 motion parallel to the Z-axis to (4,5,3)

Example 2: Suppose the current position is (1, 2, 3) and the XY-plane has been selected and the following line of NC-code is interpreted.

G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3

This calls for incremental distance mode (G91), old “Z” retract mode and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the initial Y position is 7 (=2+5), the clear Z position is 4.8 (=1.8+3) and the Z position is 4.2 (=4.8-0.6). Old Z is 3.0.

The first move is a traverse along the Z-axis to (1,2,4.8), since old Z < clear Z.

The first repeat consists of three moves.

Step #	Description
1	G00 motion parallel to the XY-plane to (5,7,4.8)
2	G01 motion parallel to the Z-axis to (5,7, 4.2)
3	G00 motion parallel to the Z-axis to (5,7,4.8)

The second repeat consists of three moves. The X position is reset to 9 (=5+4) and the Y position to 12 (=7+5).

Step # Description

1	G00 motion parallel to the XY-plane to (9,12,4.8)
2	G01 motion parallel to the Z-axis to (9,12, 4.2)
3	G00 motion parallel to the Z-axis to (9,12,4.8)

The third repeat consists of three moves. The X position is reset to 13 (=9+4) and the Y position to 17 (=12+5).

Step # Description

1	G00 motion parallel to the XY-plane to (13,17,4.8)
2	G01 motion parallel to the Z-axis to (13,17, 4.2)
3	G00 motion parallel to the Z-axis to (13,17,4.8)

Example Code using G81 Cycle:

(Sample Program G81EX18:)

(Workpiece Size: X4, Y3, Z1)

(Tool: Tool #6, 3/4" HSS DRILL)

(Tool Start Position: X0, Y0, Z1)

N2 G90 G80 G40 G54 G20 G17 G94 G64 (Safety Block)

N5 G90 G80 G20

N10 M06 T6 G43 H6

N15 M03 S1300

N20 G00 X1 Y1

N25 Z0.5

N30 G81 Z-0.25 R0.125 F5 (Drill Cycle Invoked)

N35 X2

N40 X3

N45 Y2

N50 X2

N55 X1

N60 G80 G00 Z1 (Cancel Canned Cycles)

N65 X0 Y0

N70 M05

N75 M30

PROGRAMMING

7.4.4 Simple Drilling Cycle (dwell) – G82

The G82 cycle is intended for drilling. Program: G82 X~ Y~ Z~ A~ R~ L~ P~

Step # Description	
1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate to the Z position
3	Dwell for the P number of seconds
4	Retract the Z-axis at traverse rate to clear Z

7.4.5 Peck Drilling Cycle – G83

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. See also G73. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a delta increment along the Z-axis. Program: G83 X~ Y~ Z~ A~ R~ L~ Q~

Word Definition	
Q~	This cycle takes a Q number which represents a delta increment along the Z-axis
Step # Description	
1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep
3	Rapid back out to the clear Z
4	Repeat steps 1, 2 and 3 until the Z position is reached at step 1
5	Rapid back down to the current hole bottom, backed off a bit
6	Retract the Z-axis at traverse rate to clear Z

It is an error if:

- The Q number is negative or zero

7.4.6 Boring Cycle (feedrate out) – G85

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling. Program: G85 X~ Y~ Z~ A~ R~ L~

Step # Description	
1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate to the Z position
3	Retract the Z-axis at the current feed rate to clear Z

7.4.7 Boring Cycle (dwell, rapid out) – G86

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell.

Program: G86 X~ Y~ Z~ A~ R~ L~ P~

Step # Description

1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate to the Z position
3	Dwell for the P number of seconds
4	Stop the spindle turning
5	Retract the Z-axis at traverse rate to clear Z
6	Restart the spindle in the direction it was going
7	Move the Z-axis only at the current feed rate to the Z position

The spindle must be turning before this cycle is used. It is an error if:

- The spindle is not turning before this cycle is executed

7.4.8 Boring Cycle (dwell, manual out) – G88

The G88 cycle is intended for boring and uses a P word, where P specifies the number of seconds to dwell.

Program: G88 X~ Y~ Z~ A~ R~ L~ P~

Step # Description

1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate to the Z position
3	Dwell for the P number of seconds
4	Stop the spindle turning
5	Stop the program so the operator can retract the spindle manually
6	Restart the spindle in the direction it was going

7.4.9 Boring Cycle (dwell, feedrate out) – G89

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell.

Program: G89 X~ Y~ Z~ A~ R~ L~ P~

Step # Description

1	Preliminary canned cycle motion
2	Move the Z-axis only at the current feed rate to the Z position
3	Dwell for the P number of seconds
4	Retract the Z-axis at the current feed rate to clear Z

PROGRAMMING

7.5 Built-in M-codes

M-codes interpreted directly by the operating system are detailed in the table below:

M-code	Meaning
M00	Program stop
M01	Optional program stop
M02	Program end
M03	Rotate spindle clockwise
M05	Stop spindle rotation
M07 or M08	Coolant on
M09	All coolant off
M30	Program end and rewind
M48	Enable speed and feed override
M49	Disable speed and feed override
M98	Call subroutine
M99	Return from subroutine/repeat
M100 to M199	Operator defined M-codes

7.5.1 Program Stop and Program End – M00, M01, M02, and M30

To stop a running program temporarily, regardless of the optional stop switch setting, program: M00

To stop a running program temporarily, but only if the optional stop switch is on, program: M01

It is OK to program M00 and M01 in MDI mode, but the effect probably won't be noticeable because normal behavior in MDI mode is to stop after each line of input.

If a program is stopped by an M00, M01, pressing the *Cycle Start* button restarts the program at the following line of the G-code program.

To end a program, program: M02 or M30. M02 leaves the next line to be executed as the M02 line. M30 rewinds the G-code file. These commands can have the following effects depending on the options chosen on the Configure>Logic dialog:

- Axis offsets are set to zero (like G92.2) and origin offsets are set to the default (like G54)
- Selected plane is set to XY (like G17)
- Distance mode is set to absolute (like G90)
- Feed rate mode is set to units per minute mode (like G94)
- Feed and speed overrides are set to on (like M48)
- *Cutter Compensation* is turned off (like G40)
- The spindle is stopped (like M05)
- The current motion mode is set to G01 (like G01)

- Coolant is turned off (like M09)

No more lines of code in the file are executed after the M02 or M30 command is executed. Pressing *Cycle Start* starts the program back at the beginning of the file.

7.5.2 Spindle Control – M03, M05

To start the spindle turning clockwise (forward) at the currently programmed speed, program: M03

The speed is programmed by the S word.

To stop the spindle from turning, program: M05

It is OK to use M03 if the spindle speed is set to zero; if this is done (or if the speed override switch is enabled and set to zero), the spindle won't start turning. If later the spindle speed is set above zero (or the override switch is turned up), the spindle starts turning. It is permitted to use M03 when the spindle is already turning or to use M05 when the spindle is already stopped.

7.5.3 Tool Change – M06

To execute a tool change sequence, program: M06

M06 behaves differently depending on whether or not a mill is equipped with an ATC (automatic tool changer).

Mill is not equipped with an ATC:	M06 commands the mill, stops the spindle, pauses program execution, and prompts operator to change tools by flashing <i>Tool Change</i> LED. The program resumes after the operator presses the <i>Cycle Start</i> button to confirm that the tool has been changed.
Mill is equipped with an ATC:	If the requested tool (T number) is assigned to the carousel, M06 initiates an automatic tool change. If the tool is not assigned to the carousel, the operator is prompted to manually change the tool and press <i>Cycle Start</i> to confirm the tool change. This resumes the program.

You are strongly advised to put the T~, the M06 and the G43 H~ on one line (block) of code. See G43 for more details.

Example: N191 M06 T3 G43 H3

7.5.4 Coolant Control – M07, M08, and M09

To turn coolant on, program: M07

To turn flood coolant on, program: M08

To turn all coolant off, program: M09

PROGRAMMING

It is always OK to use any of these commands, regardless of what coolant is on or off.

7.5.5 Override Control – M48, M49

To enable the speed and feed override, program: M48

To disable both overrides, program: M49

It is OK to enable or disable the switches when they are already enabled or disabled.

7.5.6 Feed Override Control – M50

To enable the feed rate override control, program: M50 P1

The P1 is optional.

To disable the feed rate control, program: M50 P0

When feed rate override control is disabled, the feed rate override slider has no influence, and all motion is executed at programmed feed rate (unless there is an adaptive feed rate override active).

7.5.7 Spindle Speed Override Control – M51

To enable the spindle speed override control, program: M51 P1

The P1 is optional.

To disable the spindle speed override control, program: M51 P0

When spindle speed override control is disabled, the spindle speed override slider has no influence, and the spindle speed is equal to the value of the S-word (see *Spindle Speed* later in this chapter).

7.5.8 Set Current Tool Number – M61

To change the current tool number while in MDI or manual mode, program: M61 Q~

Word	Definition
Q~	Tool number

One use is when you power on the system with a tool selected but the tool turret is set for a different tool to that indicated. You can set that tool number without doing a tool change operation. It is an error if: Q~ is not 0 or greater

7.5.9 Set Output State – M64, M65

NOTE: These commands are only useful when the mill is equipped with the USB I/O Module (PN 32616).

There are four output relays available on the USB I/O module.

The outputs are activated using M64 (contact close) and deactivated with M65 (contact open). There are four contacts, numbered from 0 to 3. The contact is specified by the P word.

For example:

- Activating the first relay: M64 P0
- Activating the second relay: M64 P1

The outputs are deactivated using M65 with the P word specifying the relay.

For example:

- Deactivating the second relay: M65 P1
- Deactivating the fourth relay: M65 P3

There is only one P word and one relay per line. Each relay command must be done on an individual line.

The following is legal:

```
M64 P0  
M64 P2  
M64 P3
```

The following is not legal:

```
M64 P023  
M64 P0 P2 P3
```

7.5.10 Wait on Input – M66

NOTE: These commands are only useful when the mill is equipped with the USB I/O Module (PN 32616).

There are four digital inputs available on the USB I/O module.

M66 P- | E- <L->

Word	Definition
P-	Specifies the digital input number from 0 to 3.
L-	Specifies the wait mode type: <ul style="list-style-type: none">• Mode 0: IMMEDIATE – no waiting, returns immediately. The value of the input at that time is stored in parameter #5399.• Mode 1: RISE – waits for the selected input to perform a rise event.• Mode 2: FALL – waits for the selected input to perform a fall event.• Mode 3: HIGH – waits for the selected input to go to the HIGH state.• Mode 4: LOW – waits for the selected input to go to the LOW state.
Q-	Specifies the timeout in seconds for waiting. The Q value is ignored if the L-word is zero (IMMEDIATE). A Q value of zero is an error if the L-word is non-zero.

7.6 Other Input Codes

7.6.1 Feed Rate – F

To set the feed rate, program: F~

Depending on the setting of the feed mode toggle, the rate may be in units-per-minute or units-per-rev of the spindle. The units are those defined by the G20/G21 mode. The feed rate may sometimes be overridden as described in M48 and M49 above.

7.6.2 Spindle Speed – S

To set the speed in revolutions per minute (rpm) of the spindle, program: S~

The spindle turns at the commanded speed when it has been programmed to start turning. It is OK to program an S word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100 percent, the speed is different from what is programmed. It is OK to program S0, but the spindle does turn if that is done. It is an error if:

- The S number is negative

7.6.3 Change Tool Number – T

It is the programmer's responsibility to ensure that the carriage is in a safe place for changing tools, for example by using G30. This allows optimization of motion which can save time, especially with gang tooling. A pause for manual intervention can always be provided by an M00 or M01 before the tool change. It is an error if:

- A negative T number is used or a T number larger than 54 is used

7.7 Advanced Programming with Parameters and Expressions

This section describes the parameter and expression programming language features of PathPilot. You may wish to skip this section on first reading as these features are not used in common G-code application (hand coding), G-code created by the PathPilot conversational programming interface, or the majority of third-party CAM-programming systems.

NOTE: *There are significant differences between controls in the way parameters work. Do not assume that code from another control works in the same way with the operating system. Tormach advises against writing parametric G-code as this is difficult to debug and very difficult for another operator to understand. Modern CAM virtually eliminates the need for it.*

7.7.1 Parameters

The RS274/NGC language supports parameters. Parameters are analogous to variables in other programming languages. PathPilot maintains an array of 10,320 numerical parameters. Many of them have specific uses. The parameters that are associated with fixtures are persistent over time. Other parameters are undefined when the operating system is loaded. The parameters are preserved when the interpreter is reset. Parameters 1 to 1000 can be used by the code of part-programs.

There are several types of parameters of different purpose and appearance. The only value type supported by parameters is floating-point; there are no string, Boolean or integer types in G-code like in other programming languages. However, logic expressions can be formulated with Boolean operators (AND, OR, XOR, and the comparison operators EQ, NE, GT, GE ,LT, LE), and the MOD, ROUND, FUP and FIX operators support integer arithmetic.

Parameter Syntax

There are three types of parameters, numbered, named local, and named global. The type of the parameter is defined by its syntax:

```
numbered - #4711
named local - #<localvalue>
named global - #<_globalvalue>
```

Parameter Scope

The scope of a parameter is either global or local within a subroutine. The scope of each parameter is inferred from its syntax. Subroutine parameters and named local parameters have local scope. Named global parameters and all numbered parameters starting from #31 are global in scope. RS274/NGC uses lexical scoping. In a subroutine only the local parameters defined therein, and any global parameters are visible. The local parameters of a calling procedure are not visible in a called procedure.

Behavior of Uninitialized Parameters

Uninitialized global parameters and unused subroutine parameters return the value zero when used in an expression. Uninitialized named parameters signal an error when used in an expression.

Parameter Mode

The mode of a parameter can either be read/write or read-only. Read/write parameters may be assigned values within an assignment statement. Read-only parameters cannot be assigned values. They may appear in expressions, but not on the left-hand side of an assignment statement.

Persistence and Volatility

Parameters can either be persistent or volatile. When the operating system is powered off, volatile parameters lose their values and are reset to zero. The values of persistent parameters are saved in a disc file and restored to their previous values when the operating system is powered on again. All parameters except numbered parameters in the current persistent range (5163 to 5390) are volatile.

Intended Usage

Numbered parameters in the range #31-#5000, named global, and local parameters are available for general-purpose storage of floating-point values, like intermediate results, flags, etc., throughout program execution. They are read/write (can be assigned a value). Subroutine parameters, numbered parameters #1-#30, and system parameters are read-only and not available for general use. Subroutine parameters are used to hold the actual parameters passed to a subroutine. Numbered parameters in the range of #1-#30 are used to access offsets of coordinate systems. System parameters are used to determine the current running version and are read-only.

7.7.2 Parameter Types

7.7.2.1 Numbered Parameters

A numbered parameter is recognized by the pound symbol (#) followed by an integer between 1 and 5399. The parameter is referred to by this integer, and its value is whatever number is stored in the parameter. A value is stored in a parameter with the (=) operator.

Example: #3 = 15 (set parameter 3 to 15)

A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line:

#3=6 G01 X#3

is interpreted, a straight move to a point where X = 15 occurs before the value of parameter 3 is set to 6.

The # symbol takes precedence over other operations. For example, #1+2 means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, #[1+2] does mean the value found in parameter 3.

The # character may be repeated; for example ##2 means the value of parameter whose index is the (integer) value of parameter 2. PathPilot maintains a number of read-only parameters. Only parameters for the relevant axes are maintained: (X Y Z A) for mill and (X Z) for mill. The remaining parameters for unused axes are undefined.

PROGRAMMING

Read-only Parameters	Purpose
1-30	Subroutine local parameters of call arguments. These parameters are local to the subroutine. For further information, see <i>Programming with Subroutines</i> later in this chapter
31-5000	G-code operator parameters. These parameters are global in G-code file
5061-5070	Result of G38.2 probe (X Y Z A B C U V W)
5161-5169	G28 home for (X Y Z A B C U V W)
5181-5189	G30 home for (X Y Z A B C U V W)
5210	1 if G92 offsets are active, 0 if not
5211-5219	G92 offset (X Y Z A B C U V W)
5220	Current coordinate system number 1-9 for G54 - G59.3
5221-5230	Coordinate System 1, G54 (X Y Z A B C U V W R) – R denotes XY rotation angle around Z-axis
5241-5250	Coordinate System 2, G55 (X Y Z A B C U V W R)
5261-5270	Coordinate System 3, G56 (X Y Z A B C U V W R)
5281-5290	Coordinate System 4, G57 (X Y Z A B C U V W R)
5301-5310	Coordinate System 5, G58 (X Y Z A B C U V W R)
5321-5330	Coordinate System 6, G59 (X Y Z A B C U V W R)
5341-5350	Coordinate System 7, G59.1 (X Y Z A B C U V W R)
5361-5370	Coordinate System 8, G59.2 (X Y Z A B C U V W R)
5381-5390	Coordinate System 9, G59.3 (X Y Z A B C U V W R)
5399	Result of M66 – check or wait for input
5400	Current tool number
5401-5409	Tool offset (X Y Z A B C U V W)
5410	Current tool diameter
5411	Current tool front angle
5412	Current tool back angle
5413	Current tool orientation
5420-5428	Current position including offsets in current program units (X Y Z A B C U V W)

7.7.2.2 Subroutine Parameters

Subroutine parameters are specifically reserved for call arguments. By definition, these are parameters #1-#30 and are local to the subroutine.

7.7.2.3 Named Parameters

Named parameters work like numbered parameters but are easier to read and remember. All parameter names are converted to lowercase and have spaces and tabs removed. Named parameters must be enclosed with < > marks.

#<named parameter here> is a local named parameter. By default, a named parameter is local to the scope in which it is assigned.

You can't access a local parameter outside of its subroutine. This is so two subroutines can use the same parameter names without fear of one subroutine overwriting the values in another.

#<_global named parameter here> (i.e., name starting with an underscore) is a global named parameter. They are accessible from within called subroutines and may set values within subroutines that are accessible to the caller. As far as scope is concerned, they act just like regular numeric parameters. They are not made persistent by storage in a file.

Examples:

```
Declaration of named global variable
#<_endmill_dia> = 0.049
Reference to previously declared global variable
#<_endmill_rad> = [#<_endmill_dia>/2.0]
```

NOTE: The global parameters _a, _b, _c, . . . _z are reserved for special use. Do not use these parameters.

Mixed Literal and Named Parameters

```
o100 call [0.0] [0.0] [#<_inside_cutout>-#<_endmill_dia>] [#<_zcut>] [#<_feedrate>]
```

7.7.3 Expressions

An expression is a set of characters starting with a left bracket ([) and ending with a balancing right bracket (]). Located between the brackets are numbers, parameter values, binary operators, functions, and other expressions. An expression is evaluated to produce a number. An example of an expression is:

```
[1 + acos[0] - [#3 ** [4.0/2]]]
```

All expressions on a line are evaluated when the line is read and before anything on the line is executed.

7.7.3.1 Binary Operators

Binary operators only appear inside expressions. There are three types of binary operators: mathematical, logical, and relational.

PROGRAMMING

There are four basic mathematical operations: addition (+), subtraction (-), multiplication (*), and division (/). In addition, the modulus operation (MOD) finds the remainder after division of one number by another number. The power operation (***) of raising the number on the left of the operation to the power on the right. There are three logical operations: non-exclusive or (OR), exclusive or (XOR), and logical and (AND).

The relational operators are equality (EQ), inequality (NE), strictly greater than (GT), greater than or equal to (GE), strictly less than (LT), and less than or equal to (LE).

Binary operators are divided into several groups according to their precedence.

Binary Operator	Precedence
**	1 (highest)
* / MOD	2
+ -	3
EQ NE GT GE LT LE	4
AND OR XOR	5 (lowest)

If operations in different precedence groups are strung together, operations with a higher precedence are performed before operations with a lower precedence. If an expression contains more than one operation with the same precedence, the operation on the left is performed first.

Example:

[2.0 / 3 * 1.5 - 5.5 / 11.0]

is equivalent to

[[[2.0 / 3] * 1.5] - [5.5 / 11.0]]

which is equivalent to

[1.0 - 0.5]

which is

0.5

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false, and any non-zero number is equivalent to logical true.

7.7.3.2 Functions

Available functions are shown in the table below.

Function Name	Function Result
ATAN[Y]/[X]	Four quadrant inverse tangent
ABS[arg]	Absolute value
ACOS[arg]	Inverse cosine
ASIN[arg]	Inverse sine
COS[arg]	Cosine
EXP[arg]	e raised to the given power (e^x)
FIX[arg]	Round down to integer
FUP[arg]	Round up to integer
ROUND[arg]	Round to nearest integer
LN[arg]	Base-e logarithm
SIN[arg]	Sine
SQRT[arg]	Square root
TAN[arg]	Tangent
EXISTS[arg]	Check named parameter

7.8 Programming with Subroutines

Subroutines are subprograms that are called from inside another program. The following sections discuss the structure and design of subroutine programming with PathPilot.

7.8.1 Subroutine Labels and Subroutine Keywords

Subroutines are identified in a program by a unique subroutine label. The subroutine label is the letter O followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not permitted, for example) or a string of characters surrounded by <> symbols.

Examples of valid subroutine labels include:

```
O123
O99999
O<my test code>
```

Subroutine labels may be used in any order but must be unique in a program. Each subroutine label must be followed by a subroutine keyword.

The subroutine keyword defines the action associated with the subroutine label. Valid subroutine keywords and their meanings are detailed in the following table.

Subroutine Keyword	Meaning
Sub	Begin subroutine definition
Endsub	End of subroutine definition
Call	Call the subroutine
Do/while/endwhile	Execute the subroutine while a condition is true
Repeat/endrepeat	Execute the subroutine while a condition is true
If/elseif/else/endif	Conditionally execute the subroutine
Break	Break out of a while or if/elseif statement
Continue	Skip remaining code and restart at top of while or repeat loop
Return	Return a value

7.8.1.1 Defining a Subroutine

The sub and endsub keywords are used to define the beginning and end a subroutine. All lines of code between the sub and endsub keywords are considered to be part of the subroutine.

Sub, Endsub, Call Example:

```
o100 sub
G53 G00 X0 Y0 Z0 (rapid move to machine home)
o100 endsub
...
o100 call (call the subroutine here)
M02
```

Subroutines can either be defined in the program file or in a separate file. If the subroutine is defined in the same file as the main program that calls the subroutine, it must be defined before the call statement. For instance, this is valid:

```
o100 sub
G53 G00 X0 Y0 Z0 (rapid move to machine home)
o100 endsub
...
o100 call (call the subroutine here)
M02
```

But this is not:

```
o100 call (call the subroutine here)
M02
o100 sub
G53 G00 X0 Y0 Z0 (rapid move to machine home)
o100 endsub
...
```

A subroutine can be a separate file, provided the following rules are obeyed:

- The file must be named the same as your call
- The file must include a sub and endsub in the file
- The file must be in the directory Subroutines
- The file name can include lowercase letters, numbers, dashes, and underscores only
- The file can contain only a single subroutine definition
- The file must end with the extension .nc

7.8.1.2 Calling a Subroutine

To execute a subroutine in a program, it must be called. To call a subroutine, program O~ call where ~ is the subroutine name. The subroutine name may be either a named file, a numbered file, or an expression that evaluates to a valid subroutine label.

Expression Example: o[#101+2] call

Named File Example: o<myfile> call

Numbered File Example: o123 call

Optional Arguments to O~call

O~ call takes up to 30 optional arguments, which are passed to the subroutine as #1, #2, . . . , #N. Unused parameters from #N+1 to #30 have the same value as in the calling context.

Parameters #1-#30 are local to the subroutine. On return from the subroutine, the values of parameters #1 through #30 (regardless of the number of arguments) are restored to the values they had before the call.

The following calls a subroutine with three arguments:

O~ Call Example: o200 call [1] [2] [3]

Because 1 2 3 is parsed as the number 123, the parameters must be enclosed in square brackets.

Subroutine bodies may be nested. Nested subroutines may only be called after they are defined. They may be called from other functions, and may call themselves recursively if it makes sense to do so. The maximum subroutine nesting level is 10.

Subroutines do not have return values, but they may change the value of parameters above #30 and those changes are visible to the calling G-code. Subroutines may also change the value of global named parameters.

NOTE: File names are lowercase letters only; o<MyFile> is converted to o<myfile> by the interpreter.

7.8.1.3 Conditional Subroutines

Subroutines can be conditionally executed using the if/endif or the if/else/elseif/endif keyword constructs.

if/endif

The if/endif conditional will execute a block of code following the if keyword only when the if argument evaluates to true.

If/endif Example:

```
o100 sub
(notice that the if-endif block uses a different number)
o110 if [#2 GT 5]
(some code here)
o110 endif
(some more code here)
o100 endsub
```

If/elseif/else/endif

The if/elseif/else/endif conditional will execute the block of code following the if keyword when its argument evaluates to true. If the argument evaluates to false, then the code following each elseif is executed as long as the associated elseif argument evaluates to true. If no elseif keywords are present, or if all elseif arguments evaluate to false, than the code following the else keyword is executed.

If/elseif/endif example:

```
o102 if [#2 GT 5] (if parameter #2 is greater than 5 set F100)
F100
o102 elseif [#2 LT 2] (else if parameter #2 is less than 2 set F200)
F200
o102 else (else if parameter #2 is 2 through 5 set F150)
F150
o102 endif
```

7.8.1.4 Repeating Subroutines

Subroutines can be repeated a finite number of times using the repeat/endrepeat keyword.

Repeat example:

```
(Mill 5 diagonal shapes)
G91 (Incremental mode)
o103 repeat [5]
... (insert milling code here)
G00 X1 Y1 (diagonal move to next position)
o103 endrepeat
G90 (Absolute mode)
```

7.8.1.5 Looping Subroutines

Subroutines can be looped using the Do/while or while/endwhile keyword constructs.

Do/While Loop

The Do/While loop executes a block of code once and continues to execute the code block until the while argument evaluates to *true*.

Do/While Loop Example:

```
#1 = 0 (assign parameter #1 the value of 0)
o100 do
(debug, parameter 1 = #1)
o110 if [#1 EQ 2]
#1 = 3 (assign the value of 3 to parameter #1)
(msg, #1 has been assigned the value of 3)
o100 continue (skip to start of loop)
o110 endif
(some code here)
#1 = [#1 + 1] (increment the test counter)
o100 while [#1 LT 3]
M02
```

While/endwhile

The while/endwhile repeats a set of statements an indefinite number of times, as long as the while argument evaluates to true.

While/endwhile Example:

```
(draw a sawtooth shape)
G00 X1 Y0 (move to start position)
#1 = 1 (assign parameter #1 the value of 0)
F25 (set a feed rate)
o101 while [#1 LT 10]
G01 X0
G01 Y[#1/10] X1
#1 = [#1+1] (increment the test counter)
o101 endwhile
M02 (end program)
```

The following statements cause an error message and abort the interpreter:

- A return or endsub not within a sub definition
- A label on repeat which is defined elsewhere
- A label on while which is defined elsewhere and not referring to a do
- A label on if defined elsewhere
- A undefined label on else or elseif
- A label on else, elseif or endif not pointing to a matching if
- A label on break or continue which does not point to a matching while or do
- A label on endrepeat or endwhile no referring to a corresponding while or repeat

8. ACCESSORIES

This chapter describes, in brief, the options and accessories available for the PCNC 440 mill.

8.1 Stands and Enclosures

8.1.1 Full Enclosure Kit

The PCNC 440 full enclosure kit (see **Figure 8.1**), is constructed from 16-gauge powder-coated sheet metal, and features generous 24" x 24" front opening and large 14" x 21" side openings with polycarbonate windows for high visibility.

PN	Description
35606	PCNC 440 Full Enclosure Kit



Figure 8.1

8.1.2 Stand

Welded-steel frame with 16-gauge powder-coated sheet metal (see **Figure 8.2**). Stand features 19" deep front cabinet with 24" x 24" opening and storage shelf.

PN	Description
35671	PCNC 440 Machine Stand



Figure 8.2

8.2 Coolant System Options

8.2.1 Flood Coolant Kit

The flood coolant kit has a 4-gallon tank, 115 VAC coolant pump, and all required connections (see **Figure 8.3**).

PN	Description
35719	PCNC 440 Flood Coolant Kit



Figure 8.3

ACCESSORIES

8.2.2 Spray Coolant

The Fog Buster Spray Coolant Kit is a non-fogging, non-atomizing, low-pressure sprayer with 115 VAC solenoid valve included to integrate via M7/M8/M9 program commands; requires 90 psi air supply.

PN	Description
32682	Fog Buster Spray Coolant Kit

8.2.3 Coolant Hose and Accessories

Additional coolant hose, nozzle styles, valves, manifolds, and fittings are available to design a custom flood-coolant system for any PCNC mill. Individual coolant systems are also available (see **Figure 8.4**).

PN	Description
33215	Coolant Hose Accessory Kit w/ Pliers

8.2.4 Tramp Oil Pillow

These absorbent pillows are placed inside the coolant tank to remove excess tramp oil from the coolant system.

PN	Description
31925	Floating Tramp Oil Collection Pillow



Figure 8.4

8.3 Mill Options

8.3.1 LED Spindle Light

This is a bright LED work light that mounts to the spindle nose of any PCNC mill (see **Figure 8.5**).

PN	Description
34846	LED Spindle Light Kit



Figure 8.5

8.3.2 External Contactor Kit

The optional External Contactor Kit allows for control of high-current devices (<1A) using M7/M8/M9 program commands via the controlled outlet marked *Coolant* on underside of the electrical cabinet. Rated operational current is 95 A (AC3 usage @ 380 VAC).

PN	Description
33044	External Contactor Kit

8.3.3 USB M-code I/O Interface Kit

This device allows the user to assign custom M-code commands to three optically isolated inputs and four relay contact outputs. This is useful for integrating a beacon light, auto part loader, or pneumatic vise via program command.

PN	Description
32616	USB M-code I/O Interface Kit

8.3.4 Kress Spindle Kit

This electric-die grinder serves as a companion spindle to primary mill spindle (see **Figure 8.6**). Maximum speed of 29,000 RPM; used for light milling/engraving. The horizontal mounting kit (see **Figure 8.7**) is useful for light-end work on long parts.

NOTE: Kress spindle is not compatible with flood/mist coolant.



Figure 8.6



Figure 8.7

PN	Spindle Type	Coolant Compatible
31890	Kress Companion Spindle Kit (vertical)	No
32444	Kress Companion Spindle Kit, Horizontal Adaptor	—

8.3.5 Probe

Tormach's Passive Probe (see **Figure 8.8**) is used to:

- Digitize a surface
- Probe a Z surface
- Probe X or Y surface

PN	Description	Polarity
32309	Passive Probe w/10 mm TTS Mount	Passive-Low

ACCESSORIES

Automated probing functions are available to:

- Find the center of a bore
- Find the center of a circular boss
- Find the corner of a vise

NOTE: The three adjustment set screws are 120° apart (see **Figure 8.8**).

8.3.5.1 Calibrating Probe Tip

For best results, the probe must be routinely calibrated so the center line of the probe tip is coaxial to the centerline of the PCNC mill spindle. The calibration procedure should be done:

- Prior to first use
- After tip replacement
- Periodically after extended usage

1. On the *Probe* screen, click the *Probe/ETS Setup* tab (see **Figure 8.9**).

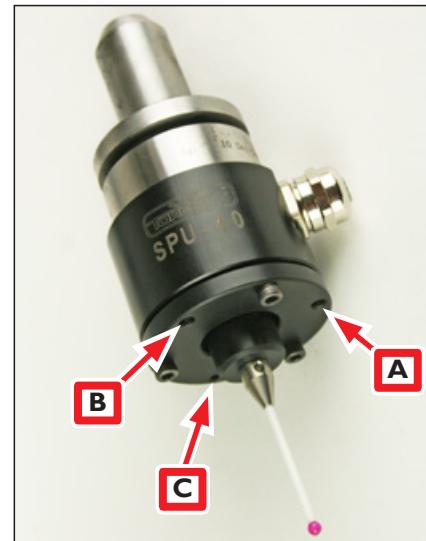


Figure 8.8

The screenshot shows the Tormach PathPilot software interface. The top navigation bar includes 'X/Y/Z Probe', 'Rect/Circ', and 'Probe/ETS Setup' (which is circled in red). Below this, there are two columns of instructions for probe setup. The left column shows a diagram of a probe being inserted into a spindle with a gauge, and the right column shows a diagram of the probe tip. The 'ETS HEIGHT' field displays '3.1496'. The bottom section contains various control buttons and status indicators, including 'CYCLE START' with options like 'SINGLE BLOCK', 'FEEDHOLD', and 'COOLANT'; 'WORK' coordinates (X: 2.9930, Y: 0.7430, Z: -0.0500, A: 0.0000) with 'DTG' values; and 'STEP' controls for feed rate (0.0001, 0.0010, 0.0100, 0.1000) and spindle speed (3500 RPM). The 'Probe' tab is highlighted with a red oval at the bottom.

Figure 8.9

2. Refer to the following *Probe Tip Adjustment* screen instructions:

- a. Orient the probe in the spindle so one of three adjustment screws is opposite the machine column (see **Figure 8.10**). Label this Screw A.
- b. Press $Y+$ button next to A.
- c. Rotate spindle 120° clockwise (as viewed from above the probe) so next screw is opposite machine column. Press $Y+$ button next to B.
- d. Similarly, rotate spindle a third time until final screw is opposite machine column. Press $Y+$ button next to C.
- e. Tighten the screw corresponding to the largest DRO value (A, B, or C). Alternatively, if the screw cannot be tightened, loosen the other two screws.
- f. Iterate this process until all DROs read the same value. All screws should be tight.

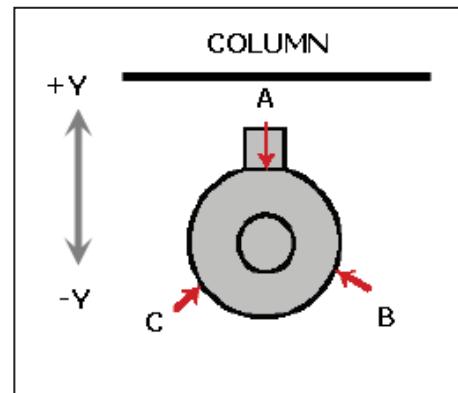


Figure 8.10

After calibration, mark the spindle pulley with a permanent marker or paint pen at a location that corresponds to the angular position of the probe cord. This allows the probe to be removed and replaced in the exact spindle orientation it was calibrated for – and eliminate error stackup.

8.3.5.2 Measuring Probe Tip Diameter

Use a micrometer to directly measure probe tip diameter. Enter this data into tool table for Tool #99.

MAINTENANCE

9. MAINTENANCE

9.1 Regular Maintenance

Scheduled maintenance intervals are detailed in the table below.

Mill Maintenance Schedule

Frequency	Completed	Item
Daily		Check oiler level and top off as needed (0.25 quart capacity)
		Retract and release manual oiler plunger each time mill is powered on and after every four hours of operation
		Clean chips from ways, carriage, and bellows' covers
		Spray exposed, non-painted metal surfaces with WD-40® or similar to prevent rust
Weekly		Check drawbar for wear; grease and adjust if needed
		Check air gauge/regulator for proper PSI (90 PSI minimum)
		Use mild cleaner to clean all exterior surfaces (no solvents)
Monthly		Inspect electrical cabinet vent and fan filter for dust build-up and wipe with a clean cloth if necessary; if dust accumulation is excessive, remove with compressed air
		Pull back way covers and inspect ways and ball screw for proper lubrication (on all axes)
Every Six Months		Inspect spindle belt for nicks, fraying, or other noticeable signs of wear
		Inspect way covers as needed to ensure proper operation; replace as needed
		Inspect oil system for blockages; clean/replace as needed
		Check lubrication hoses for signs of wear or cracking; replace as needed
		Check X-axis flex conduit for signs of wear or cracking; replace as needed
Every 12 Months		Check door switch for proper function; replace as needed

9.1.1 Rust Prevention

Exposed iron and steel surfaces will rust if proper care is not taken to protect them. The following recommendations will slow or reduce the onset of surface rust.

- If possible, install the PCNC mill in a temperature- and humidity-controlled environment.
- Always use a flood coolant recommended for machining; never use water or a coolant that does not contain rust inhibitors. When using a water-based coolant, always mix the coolant concentrate to the dilution ratio recommended by the coolant manufacturer.
- If the mill is not used for more than 72 hours, apply a light mist of water repellent oil such as WD-40 to the exposed bare metal surfaces.
- Apply way oil or machine oil directly to the table surface under trapped areas like a vise or fixture on the mill. Light surface rust on the table can be removed with a machinist's stone.

9.1.2 Way Covers

Way covers serve an important function – they keep chips and abrasive debris from damaging the slideways. Clean and inspect the way covers per maintenance schedule.

9.1.3 Flood Coolant System

Regular maintenance of the flood coolant system will prolong the service life of the coolant pump.

- Collect tramp oil in the tank with an absorbent pillow or a mechanical oil skimmer. Replace pillows as needed.

PN	Description
31925	Floating Tramp Oil Collection Pillow
35244	Skimpy Oil Skimmer Kit

- Coolant can scum if allowed to sit for a prolonged period; replace as needed.
- Check the impeller for obstructions.
- Clean coolant reservoir regularly.

NOTE: Check with local authorities on proper handling and disposal of new and used coolant.

9.1.4 Lubrication System

The lubrication system distributes oil to 15 points throughout the mill. This includes the 12 sliding surfaces (four each on the three axes) and three ball screw nuts – some of the most critical and expensive mechanical parts of the mill. Any dirt or foreign material suspended in the oil is going to be delivered directly to these parts and can dramatically shorten the operational service of the mill.

- Use only new, high quality ISO VG 68 grade Machine Oil.

PN	Description
31386	Tormach WL-68

MAINTENANCE

- Alternative choices include Perkins Perlube WL-68, Tonna 68 (Shell), Vactra No. 2 (Mobil), Way-lube 68 (Sunoco), WayLube 68 (Texaco), Febis 68 (Esso) or equivalent oil.
- Clean off the cover and surrounding area to remove debris before refilling the oiler reservoir.
- Periodically inspect way surfaces and ball screws to confirm a proper oil film is present. Absence of an oil film can be an indicator of clog oil lines or fittings.

The X, Y, and Z slideways have a thin layer of PTFE-filled acetyl plastic bonded to each sliding surface. The material is commonly known under the trade names of Turcite® or Rulon®.

This is state of the art technology for oil lubricated slideways and superior to plain ground surfaces or hardened and chromed surfaces. No data is available on how long the material will last on the PCNC, but there have been no reports of appreciable wear, even on mills that are reported to have seen more than 5000 hours of operation.

If you use the lubrication system and keep the protective bellows in good shape, the slideways are not maintenance items.

A shot of lubrication should be given after every four hours of operation and after the mill has stood unused for 48 hours or longer.

If the mill's lubrication system becomes clogged, brass oil system fittings can be cleaned by soaking them overnight in a degreaser or solvent like WD-40. Replace clogged plastic oil line tubing.

9.1.4.1 Manual Oiler Specifics

- The manual pump is spring loaded. Retract and release the plunger and the spring force creates light hydraulic pressure to push oil through the lines. You can get the oil out quicker by pushing the plunger a bit, but too much force can pop off oil lines. A shot of lubrication should be given after every four hours of operation and each time the mill is powered on.
- You will have a more uniform distribution of oil if the mill is moving when the hydraulic pressure is applied.
- The manual oiler draws oil from the reservoir on the pull stroke and delivers it to the mill on the push stroke. If at some point the manual oiler seems much easier on the push stroke, make certain that you do not have a broken oil line.
- Extreme axis positions can expose the lubrication system channels that are cut into the way support saddle surfaces. If the pump is used in those positions, the hydraulic force of the oil will not apply it throughout the mill as intended. Instead, the oil will simply squirt out at the point where the oil channel is exposed.
- After a long period of inactivity, the lubrication system may need several pumps as oil may have drained from the Z-axis lines.

9.1.5 Drawbar and TTS Collet

The drawbar, drawbar bushing, and TTS collet are wear items and should be replaced regularly. Proper lubrication and maintenance of the drawbar, drawbar bushing, and TTS collet will maximize tool holding force and prolong the service life of these components.

- Clean tool holder shanks as needed with a degreaser
- Ensure inside of collet is clean and dry; remove preservative shipping oil when first received
- Lubricate outside of collet and inside of spindle taper with Anti-seize Grease (PN 31273). Use sparingly to avoid risk of the lubricant migrating to the inside of the collet. Only the first inch of the spindle taper needs to be lubricated; remove excess lubricant with a degreaser.

PN	Description
31273	Anti-seize Grease

- Lubricate the threaded section of the drawbar, the thrust shoulder of the drawbar, and the drawbar bushing with Anti-seize Grease (PN 31273)
- Do not overtighten the drawbar; the recommended drawbar torque is 30 ft-lbs, and exceeding 40 ft-lbs of torque will reduce the operational service of the collet and drawbar
- Visually inspect the drawbar, drawbar bushing, and collet for signs of wear such as damaged or galled threads and replace as needed; it is recommended that these be replaced as a set

9.2 Spindle Belt

Inspect the spindle belt as indicated in the maintenance schedule. Replace if necessary.

PN	Description
35799	PCNC 440 Spindle Belt

9.3 Advanced Maintenance

9.3.1 Overview

Each PCNC 440 is tuned at the factory to meet or exceed certain precision metrics. These metrics are indicated on the *Certificate of Inspection* included with each mill, along with the actual values measured for each metric as part of Tormach's Quality Assurance program. The following advanced maintenance procedures may become necessary over the ownership lifetime to maintain the original factory precision:

- Gib adjustment
- Angular contact bearing preload adjustment
- Geometry adjustment (tram)

These adjustments are generally used to address component wear-in over time, but may also be needed to correct misalignment resulting from misuse, a hard crash of the system, or when some components are removed or replaced due to damage. The frequency of these procedures depends on both how the mill is used and how often.

The adjustments in this section should not be considered lightly as a wrong adjustment can adversely affect mill precision. Before making any of the adjustments in this section, consider the following:

- Why am I making this adjustment?
- How will I measure the effect of this adjustment?
- What unintended consequences may result from this adjustment?

If you do decide to make an adjustment, do not assume where the error is from. The error could be attributed to a specific problem, or from the combined effect of several problems. Mistakenly making the wrong adjustment can make matters worse. As a practical example, if the Z-axis gib is too loose, it will cause the spindle head to tilt slightly downward toward the column. It would be fairly easy to incorrectly assume that the issue is with the column and base connection (often referred to as tram), and make an adjustment by inserting shims between the column and base. Instead of correcting the real issue, this adjustment causes the column to slant back to correct for the head leaning down. Now, the mill is running in a slight parallelogram in addition to a loose head.

9.3.2 Definitions

The following definitions are important to the advanced maintenance discussion.

Lost Motion

Lost motion is the difference between commanded motion and observed motion. This is sometimes referred to as apparent backlash. There are a number of components of lost motion, including conventional backlash, bearing compressibility, sliding friction, and thermal expansion.

Backlash

Backlash is the major component of lost motion in a machine tool axis. It results from the clearance between moving mechanisms. This is sometimes referred to as play. There are two sources of conventional backlash that can be adjusted on the PCNC:

- Space between gib and way needed to support an oil film. This is tuned by tightening the gib.
- The space between the ball bearings and races in the angular contact bearing pair that supports the ball screw. This is tuned by increasing the angular contact bearing pair preload.

9.3.2.1 How to Measure Lost Motion

Correctly measuring lost motion is critical to successfully undertaking any of advanced maintenance procedures detailed in the following sections. Mill setup and tuning is done under no-load conditions. The precision measurements recorded in the *Certificate of Inspection* are taken under no-load conditions. The accuracy of a machined feature is not an indicator of machine precision.

Tool flex, workpiece flex, fixture flex, thermal expansion, and other factors contribute to the overall machined-part accuracy.

The following tools are essential:

- Dial indicator
- Dial test indicator
- Magnetic dial stand

The following method describes the proper procedure to measure X-axis backlash. An analogous procedure is used to measure Y- and Z-axis backlash.

1. Mount a dial indicator to the mill table along the X-axis to the left of the spindle, with the tip pointing at the spindle.

NOTE: If your indicator only reads in increments of .001", then the best you can hope for a reading is +/- .0005". For best results, use an indicator that has increments less than .001".

2. Jog the Y- and Z-axis to position the spindle head so the indicator tip contacts the outer diameter of the spindle cartridge, as shown in **Figure 9.1**. It is best to use a flat tip on the indicator as it will be contacting a curved surface.
3. Carefully jog the X-axis in the positive direction until the indicator contacts the spindle. After initial contact, continue jogging the X-axis in the positive direction so that the dial makes at least one complete revolution and stop jogging when the dial reads 0.
4. Zero the X DRO on PathPilot® interface.
5. In the MDI field, program a positive X move of .01" at a feed rate of 5 IPM: G01 X.01 F5. The spindle head moves slightly in the +X direction. When finished, the indicator should read .010".
6. Program an X move back to 0: G01 X0 F5. The spindle head moves slightly in the -X direction. The X DRO should say 0; however, the dial indicator should read a number very close to 0. This value is the measured lost motion.



Figure 9.1

MAINTENANCE

9.3.3 Gib Adjustment

9.3.3.1 Overview

PCNC mills use dovetail-gibbed ways to guide the X-, Y-, and Z-axis motion. Over time, the dovetail ways and gibbs wear from sliding friction and it may be necessary to tighten the gib to reduce axis backlash. To compensate for wear, the design of a dovetail-gibbed way allows for the position of the gib to be adjusted to maintain an appropriate sliding clearance.

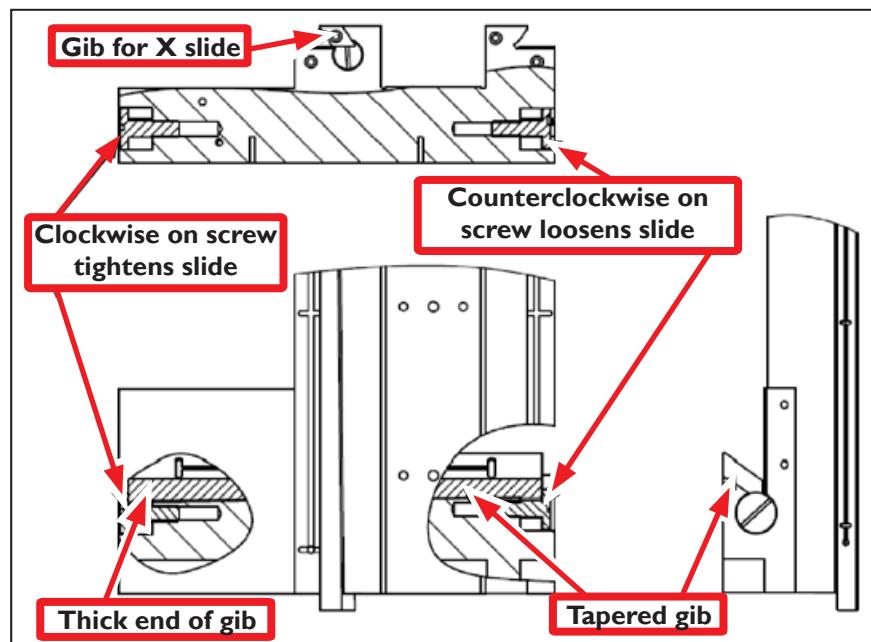


Figure 9.2

A properly adjusted gib minimizes lost motion by balancing conventional backlash and sliding friction. A gib that is too loose results in excessive conventional backlash; a gib that is too tight cannot adequately support an oil film resulting in excessive sliding friction.

The position of the tapered gib plate is controlled by two screws on either end of the gib that capture the position of the gib with respect to the saddle. These screws can be adjusted (as a pair) to tune the tightness and sliding friction of the dovetail way for each axis. The Y-axis gib mechanical detail is shown in **Figure 9.2**; both the X- and Z-axis have similar detail.

9.3.3.2 Adjustment Procedure

The gib tightening adjustment procedure for each linear axis is detailed in the table below.

Gib Plate	Tighten ¹	Notes
X-axis	Left screw clockwise/right screw counterclockwise	No cover removal required
Y-axis	Front screw clockwise/rear screw counterclockwise	Remove front and rear way covers to access gib screws
Z-axis	Upper screw clockwise/lower screw counterclockwise	Support spindle head with wooden block on table

¹To loosen, reverse rotation direction indicated in table.

NOTE: It is difficult to assess the correct clearance for the gib, as a very small adjustment can create a dramatic change in sliding friction.

The recommended method for gib adjustment is to measure axis lost motion while incrementally tightening the gib to arrive at the correct setting. The following procedure describes this method for the Z-axis. A similar procure can be used to adjust the X-axis and Y-axis gib; however, it should be noted that the X- and Y-axis gib adjustments cannot be considered in isolation. Tightening or loosening a gib on either axis also has an effect on the opposing axis.

1. Loosen upper gib screw eight rotations and tighten the lower gib screw eight rotations. This ensures gib clearance is quite loose.
2. Use a dial indicator to measure lost motion in the Z-axis (for more information, refer to *How to Measure Lost Motion* section earlier in this chapter). With a very loose gib, the majority of the measured lost motion is attributable to the backlash in the angular contact bearing pair. On a new mill, this value measured should be less than 0.002" on the Z-axis and less than .0018" on both the X- and Y-axis.
3. Tighten the gib by one turn by loosening the lower screw first, then tightening the upper screw. Measure the backlash again.
4. Repeat this procedure until the measured backlash begins to increase. At this point, the gib setting is slightly too tight.
5. Back the adjustment off to the point just before you saw the increased backlash. That is the ideal setting for the axis (see **Figure 9.3**).

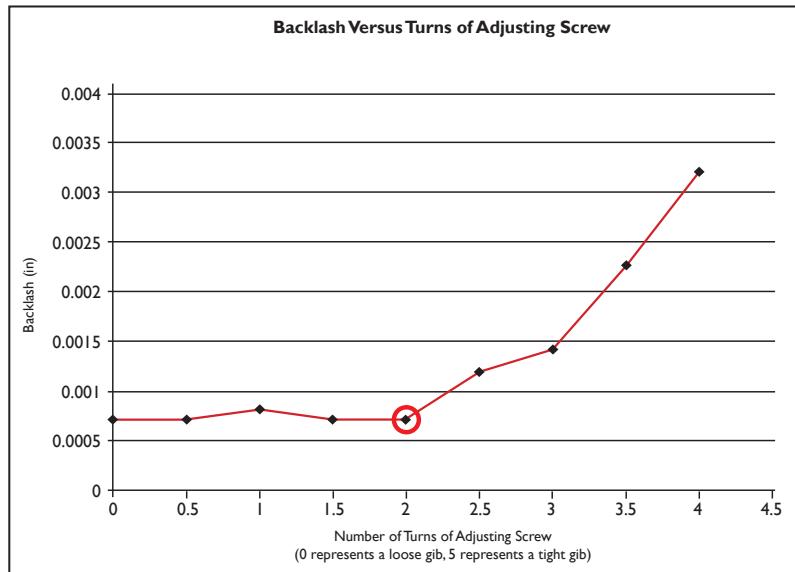


Figure 9.3

NOTE: After any gib adjustment, ensure that both adjustment screws are tight or the gib may move out of adjustment.

9.3.4 Angular Contact Bearing Preload Adjustment

9.3.4.1 Overview

Each axis utilizes a double-nut, pre-loaded ball screw. The preload in the ball nut is set at the factory by placing a precision ground spacer between the two ball nuts. Lost motion attributable to the ball screw assembly is 0.00". Ball nut preload is not operator-adjustable.

The ball screw mount bearings are located in the stepper motor for each ball screw. These are a pre-loaded angular contact bearing pair and are operator-adjustable. Under typical use, these bearings should be adjusted so that observable lost motion is between 0.0012" to 0.0018".

MAINTENANCE

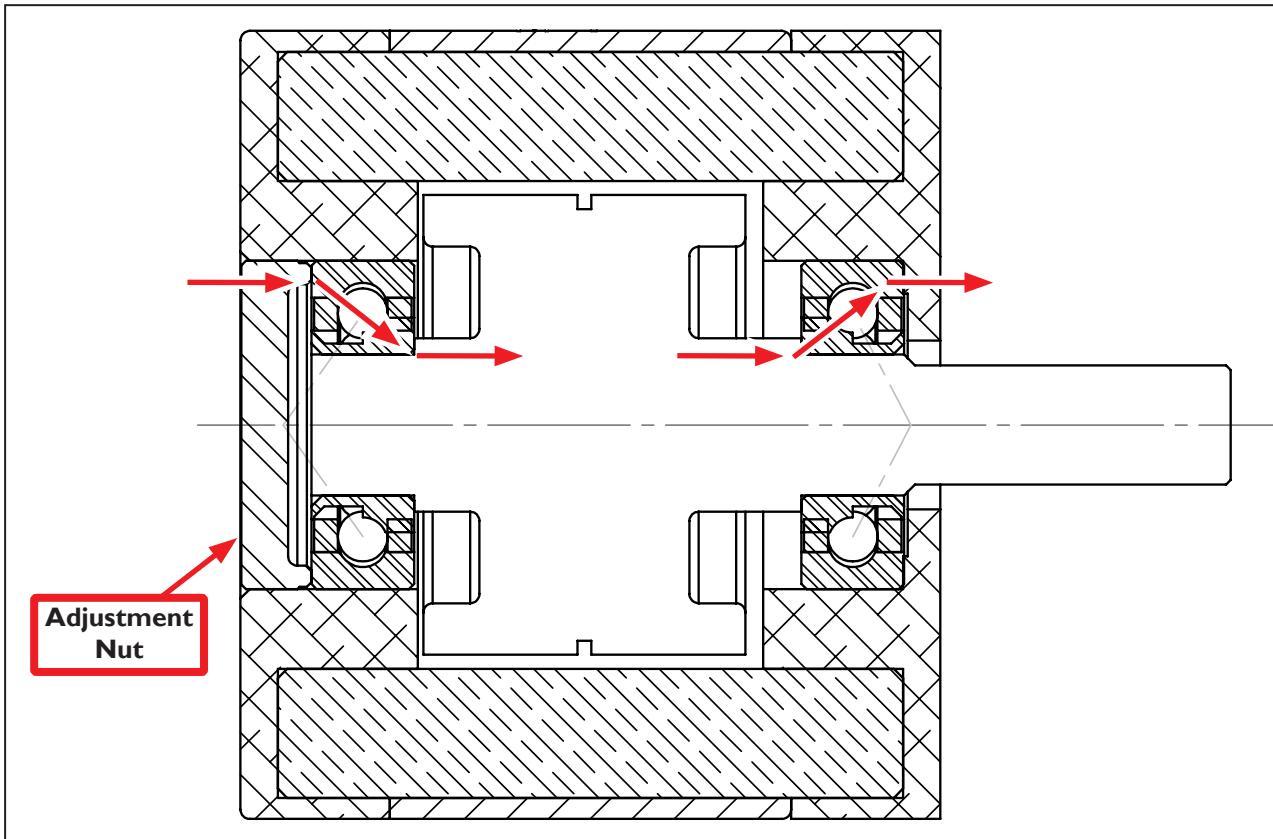


Figure 9.4

Figure 9.4 shows a cross section of the stepper motor. There are two angular contact ball bearings, forming a pre-loaded pair. The adjustment nut and stepper motor frame hold the two outer races together.

The inner races are held between the sleeve (left side) and the shoulder cut into the ball screw shaft (right side). The sleeve is held against the left inner bearing race by the adjustment nut and a lock nut. When the adjustment nut is screwed tighter toward the bearing pair, the preload increases.

Over time, it may become necessary to adjust the ball screw bearing preload to account for bearing wear. Improper ball screw bearing preload will result in either excessive backlash in the mill if it is too loose, or rapid wear and excessive friction if it is too tight. It should be noted that if your ball screw, ball nut, or angular contact bearings are worn, or if your gibbs are adjusted too tight, you will not achieve appropriate lost motion values.

9.3.4.2 Adjustment Procedure

To adjust the angular contact bearing pair preload, the following tool is required:

PN	Description
32655	Pin Spanner Wrench

The Adjustment Nut is located on the back of each stepper motor (see **Figure 9.4**).

1. Mark the current position of the adjustment nut on the back of the stepper motor.
2. Measure the lost motion in the axis. For more information, refer to *How to Measure Lost Motion* section earlier in this chapter.
3. Using a pin spanner wrench, turn the adjustment nut clockwise about 10° and measure the lost motion in the axis again. If the lost motion value decreases, adjust another 10° and re-measure. Repeat this step until the lost motion value does not decrease from your changes.

IMPORTANT! *Do not overtighten the adjustment nut; this can cause the stepper motors to stall or may damage the bearings and the motor.*

9.3.5 Geometry Adjustment of Precision Mating Surfaces

All precision mating surfaces are pinned together with tapered dowels during assembly at the factory. The pinned connection ensures that factory alignment is maintained in the event of a tool crash. Each dowel pin has a small metric threaded hole in the center that can be used to extract the dowel should it be required for disassembly.

Under typical usage, no adjustment of pinned connections should be necessary. In the event of a hard crash, shims can be used to make minor alignment adjustments between pinned components. Small adjustments (less than .010") will generally not require a full disassembly of the pinned connection. In these cases, the alignment dowels can be left in place, and the shims can be inserted into a small opening created by loosening the bolted connections.

9.4 Spindle Bearing Adjustment

During operation, sustained high spindle speed will bring the spindle bearings to about 155°F (68°C). This is a normal condition. Spindle bearing preload is set at the factory and is not operator-adjustable.

IMPORTANT! *Do not attempt to adjust spindle bearing preload. Failure to do so will adversely impact spindle balance.*

9.5 Transporting the Mill

Follow these steps when preparing the mill for transport to a new location:

1. Place a solid block of wood on the table underneath the spindle and lower the Z-axis so that the wood is slightly compressed.
2. Apply a light mist of water repellent oil, such as WD-40, to all exposed bare metal surfaces before moving the mill to protect against corrosion.
3. Remove the PCNC 440 from the stand and secure to a shipping pallet for vehicle transport.

MAINTENANCE

IMPORTANT! *The PCNC 440 must be removed from the Tormach stand (if equipped) for transport. The stand is not designed to support the weight of the mill during transport.*

Warranty is void if the mill is disassembled. Tormach recognizes that there are situations where operators need to disassemble their mill and has made provisions in the design of the mill to facilitate this. Nevertheless, Tormach cannot be held responsible for alignment, precision, and operating functions after the mill has been disassembled. Test your mill before disassembling it.

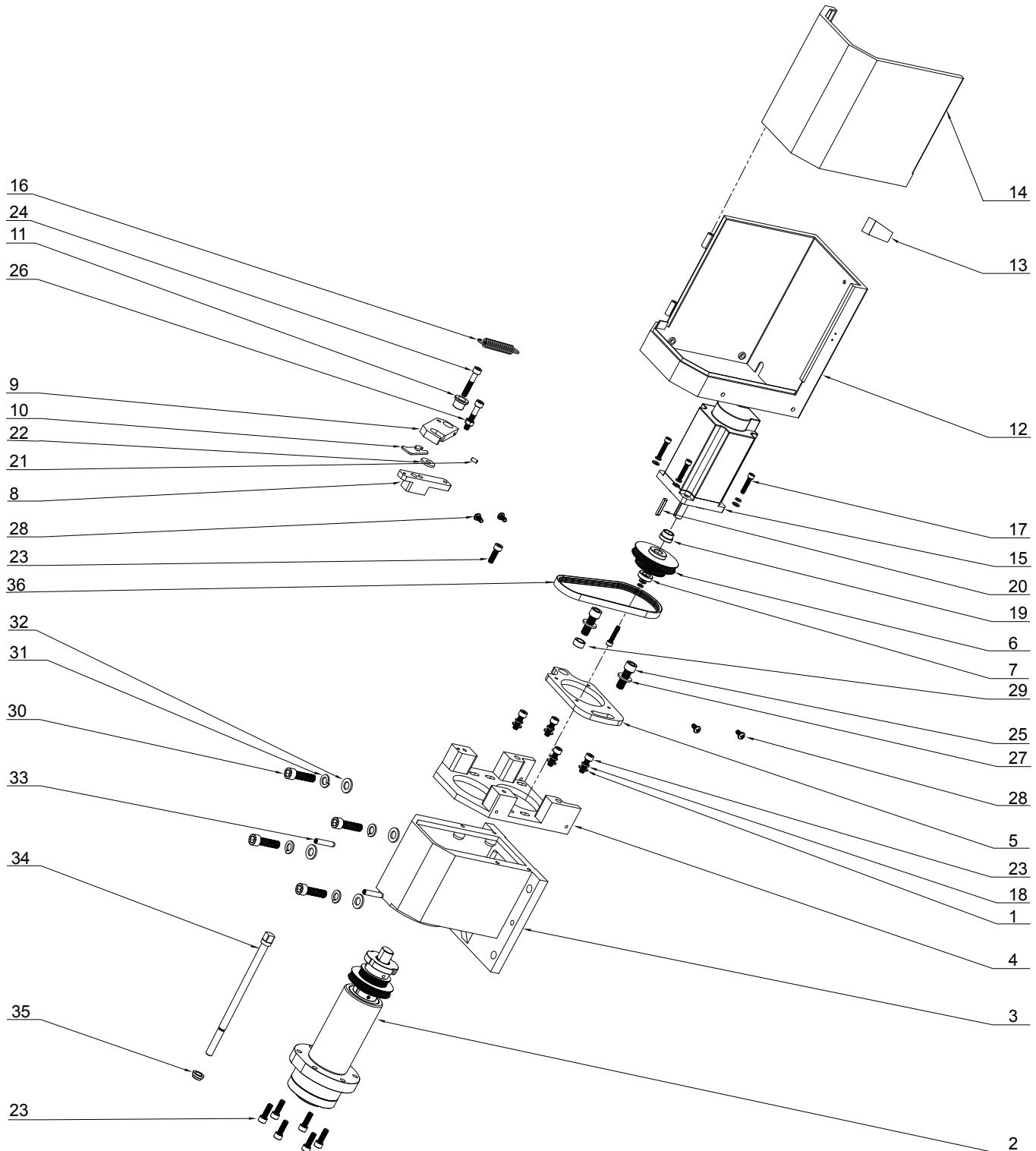
The major sub-assemblies of spindle head, column, and base are bolted and dowelled together so the mill can be separated into smaller components to meet very challenging transport problems. Note, however, that this entails disconnecting wiring and the lubrication lines. We recommend taking photographs from all angles, including detailed photos of any wires or oil lines that will be taken apart. Dowel pins must be removed before the bolts on disassembly. Dowel pins must be installed before the bolts on re-assembly.

Contact Tormach Technical Support if more detailed advice is required.

DIAGRAMS AND PARTS LIST

10. DIAGRAMS AND PARTS LIST

10.1 Spindle Head Assembly (exploded view)



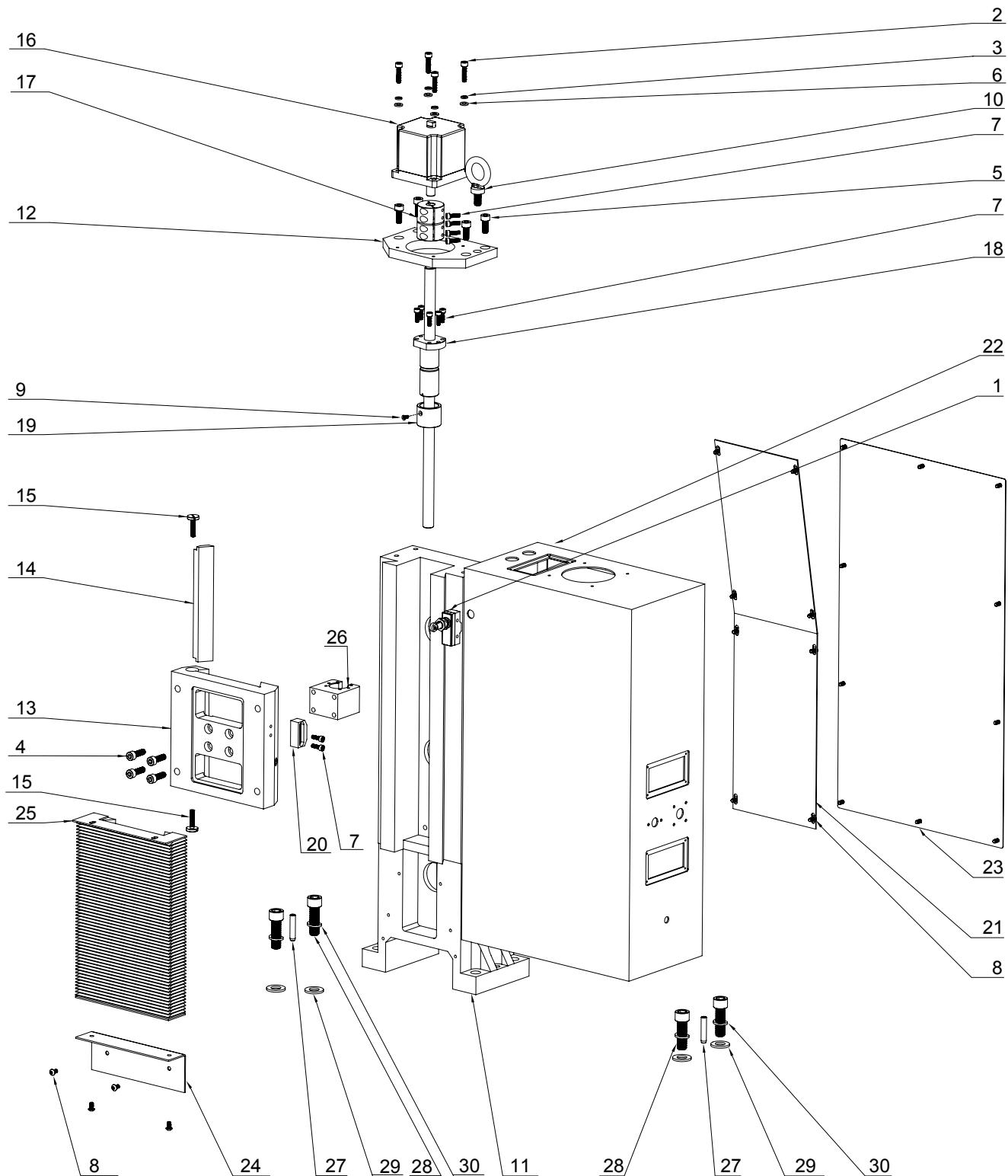
DIAGRAMS AND PARTS LIST

Spindle Head Assembly Parts List

ID	PN	Description
1	30531	M8 Flat Washer
2	33163	Spindle Assembly
3	35741	Spindle Head
4	35742	Spindle Motor Mount
5	35743	Motor Pivot Plate
6	35744	Spindle Motor Pulley
7	35745	Pulley Washer
8	35746	Spindle Lock Block
9	35747	Spindle Lock
10	35748	Door Jamb
11	35749	Jamb Bushing
12	35750	Spindle Motor Cover
13	37312	Spindle Cover Latch
14	35751	Motor Cover Door
15	35422	Spindle Motor
16	35753	Spindle Lock Spring
17	35754	M6 x 30 mm Socket Head Cap Screw
18	35755	Lock Washer
19	35756	Spindle Motor Pulley Bushing
20	35757	Spindle Motor Key
21	35758	M5 x 10 mm Dowel
22	35759	Plastic Washer
23	35760	M8 x 25 mm Socket Head Cap Screw
24	35761	M8 x 45 mm Socket Head Cap Screw
25	35762	M12 x 35 mm Socket Head Cap Screw
26	35763	M8 Hex Nut
27	35764	M12 Flat Washer
28	35765	M6 x 12 mm Phillips Head Machine Screw
29	35766	Spindle Motor Plate Bushing
30	35801	M12 x 45 mm Socket Head Cap Screw
31	35802	M12 Split Lock Washer
32	35803	M12 Flat Washer
33	35804	M8 x 40 mm Pull Dowel
34	33014	Drawbar
35	33168	Drawbar Collar
36	35799	Spindle Belt (Gates # 170J-4)

DIAGRAMS AND PARTS LIST

10.2 Z-axis Column Assembly (exploded view)



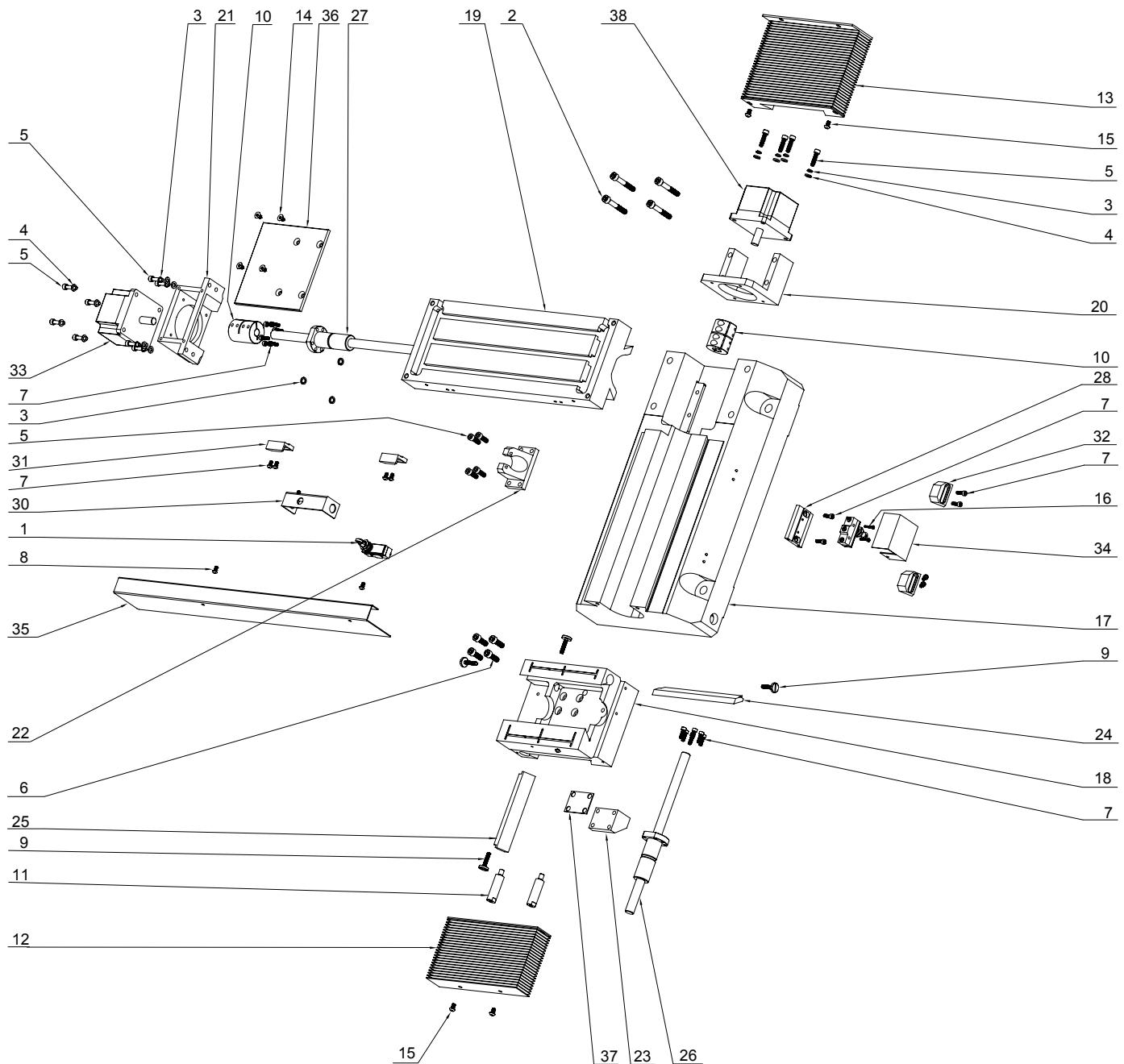
DIAGRAMS AND PARTS LIST

Z-axis Column Assembly Parts List

ID	PN	Description
1	31860	Limit Switch
2	31685	M6 x 25 mm Socket Head Cap Screw
3	35755	Lock Washer
4	35760	M8 x 25 mm Socket Head Cap Screw
5	35771	M8 x 20 mm Socket Head Cap Screw
6	35772	M6 Flat Washer
7	35773	M5 x 16 mm Socket Head Cap Screw
8	35774	M5 x 10 mm Button Head Cap Screw
9	35775	M3 x 6 mm Flat Head Cap Screw
10	35776	M10 Eyebolt
11	35777	Z-column
12	35778	Z-motor Plate
13	35779	Z-saddle
14	35780	Z-gib
15	35781	Gib Screw
16	35782	Z-axis Motor Assembly
17	35783	Z-coupling .5"-16 mm
18	35784	Z-axis 16 mm Ballscrew and Nut
19	35785	Z-ballscrew Bumper
20	35786	Z-switch Dog
21	35787	Z-column Cover
22	35788	Electrical Cabinet
23	35789	Electrical Cabinet Cover
24	35790	Z-bellows Bracket
25	35791	Z-bellows
26	35792	Z-ballnut Bracket
27	35804	M8 x 40 mm Pull Dowel
28	35805	M14 x 45 mm Socket Head Cap Screw
29	35806	M14 Flat Washer
30	35807	M14 Split Lock Washer

DIAGRAMS AND PARTS LIST

10.3 X-Y Axis Assembly (exploded view)



DIAGRAMS AND PARTS LIST

X-Y Axis Assembly Parts List

ID	PN	Description
1	31860	Limit Switch
2	30970	M8 x 55 mm Socket Head Cap Screw
3	31379	M6 Lock Washer
4	31513	M6 Flat Washer
5	31543	M6 x 25 mm Socket Head Cap Screw
6	35760	M8 x 25 mm Socket Head Cap Screw
7	35773	M5 x 16 mm Socket Head Cap Screw
8	35774	M5 x 10 mm Button Head Cap Screw
9	35781	Gib Screw
10	35783	Coupling .5"-16 mm
11	35793	Bellows Stand Off
12	35794	Y-front Bellows
13	35795	Y-rear Bellows
14	35796	M6 x 10 mm Flat Head Cap Screw
15	35797	M6 x 10 mm Button Head Cap Screw
16	35798	M3 x 20 mm Pan Head Machine Screw
17	35808	Machine Base
18	35809	X-Y Saddle
19	35810	Machine Table
20	35811	Y-motor Mount
21	35812	X-motor Mount
22	35813	X-ballnut Bracket
23	35814	Y-ballnut Bracket
24	35815	X-gib
25	35816	Y-gib
26	35818	Y-axis 16 mm Ballscrew and Nut
27	35817	X-axis 16 mm Ballscrew and Nut
28	35819	Y-switch Plate
29	35820	Y-switch Bracket
30	35821	X-switch Bracket
31	35822	X-switch Dog
32	35823	Y-switch Dog
33	35824	X-axis Motor Assembly
34	35825	Y-switch Cover
35	35826	Front Drip Guard
36	35827	Table Tray
37	35828	Y-ballnut Shim

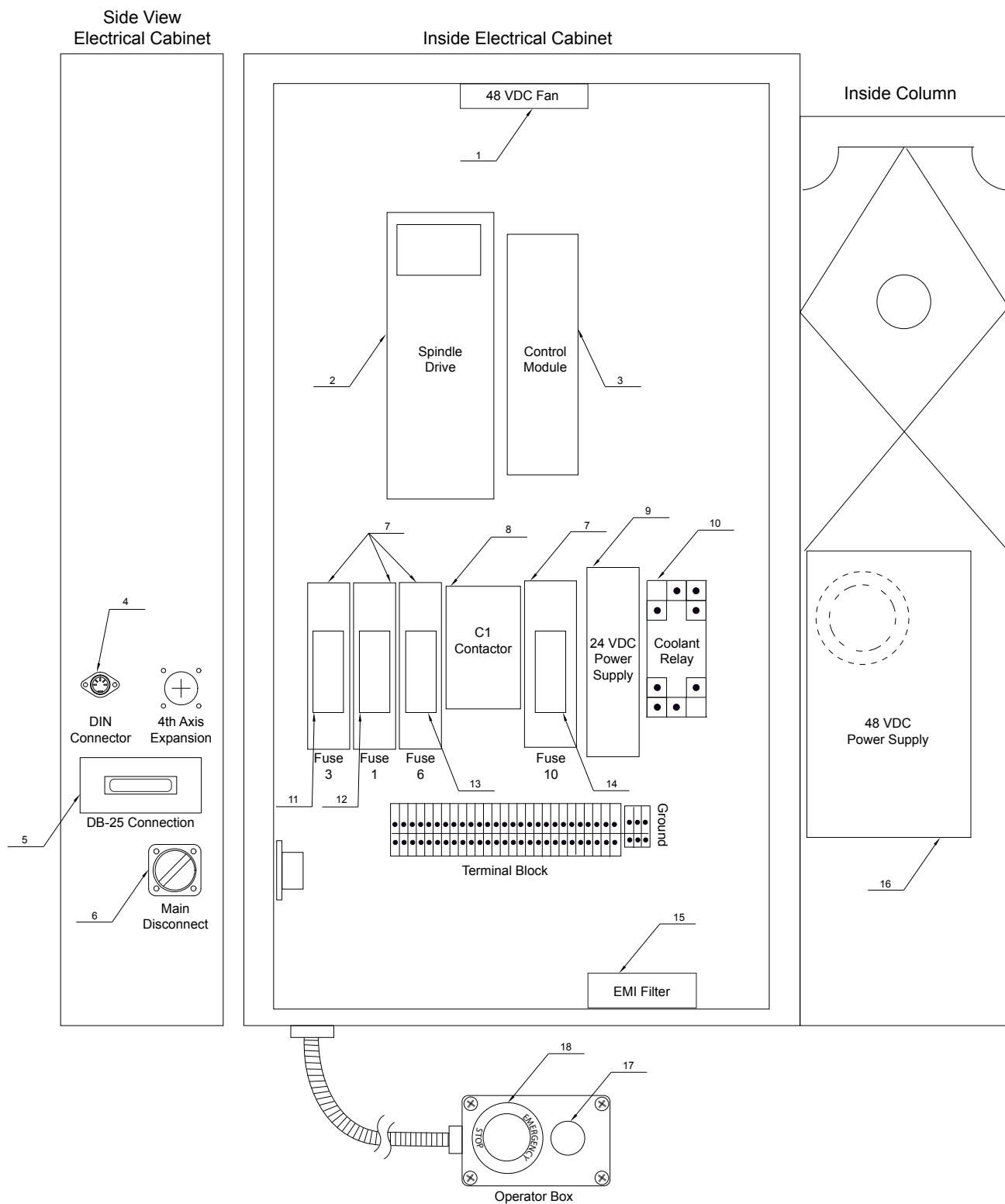
DIAGRAMS AND PARTS LIST

X-Y Axis Assembly Parts List

ID	PN	Description
38	35423	Y-axis Motor Assembly

DIAGRAMS AND PARTS LIST

10.4 Electrical Cabinet



DIAGRAMS AND PARTS LIST

Electrical Cabinet Parts List

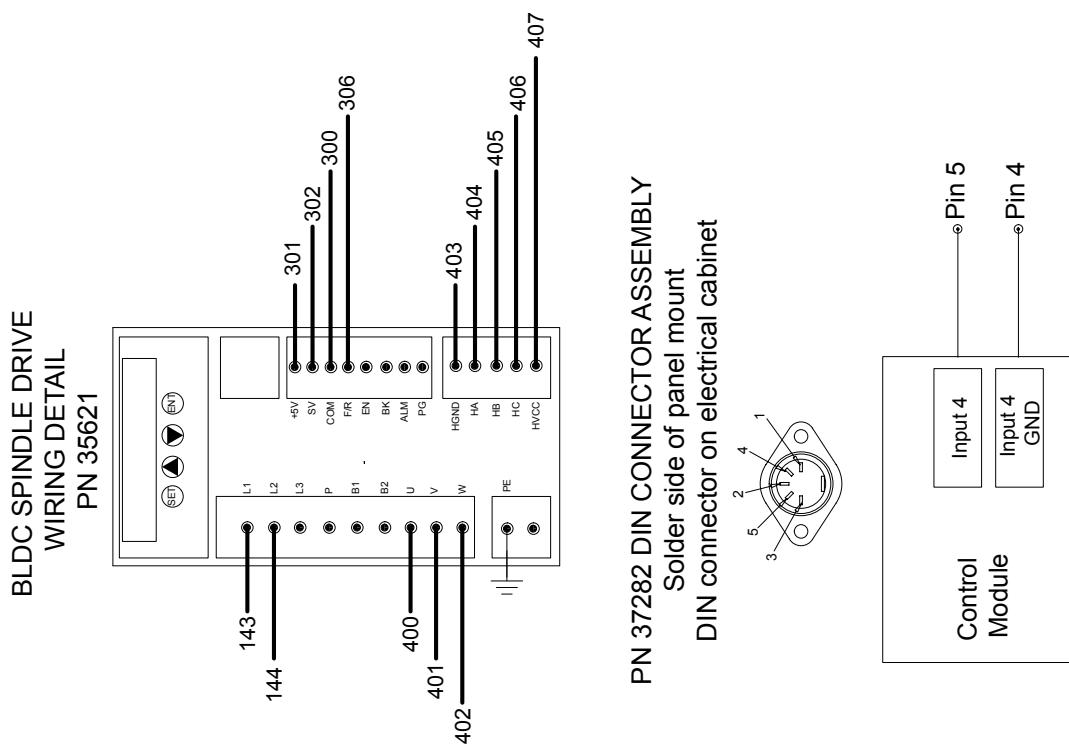
ID	PN	Description
1	35629	48 VDC Fan
2	35621	Spindle Drive
3	33374	Control Module
4	37282	DIN Connector Assembly
5	35721	DB-25 Ribbon Cable Assembly
6	35720	Main Disconnect
7	30511	Fuse Holder
8	31659	C1 Contactor
9	37153	24 VDC Power Supply
10	37154	Coolant Relay
11	35627	Fuse 3 – 2A
12	30456	Fuse 1 – 6A
13	30455	Fuse 6 – 15A
14	37283	Fuse 10 – 10A
15	31440	EMI Filter
16	35628	48 VDC Power Supply
17	33219	Push Button
18	35980	E-stop
—	30627	16 mm Flex Conduit ¹
—	30722	12 mm Flex Conduit ²
—	30628	Connector for 16 mm Flex Conduit
—	30723	Connector for 12 mm Flex Conduit

¹16 mm flex conduit used for: X-axis motor, Y-axis motor, Z-axis motor, spindle motor, E-stop, and X-Y limit switch

²12 mm flex conduit used for Y-axis limit switch

DIAGRAMS AND PARTS LIST

10.5 Connections



DIAGRAMS AND PARTS LIST

10.6 Stepper Connections

Stepper Motors to Stepper Driver			
Motor Lead	Wire Number	Driver Pin	Axis
RED	308	A+	X
GRN	309	A-	X
YEL	310	B+	X
BLU	311	B-	X
RED	312	A+	Y
GRN	313	A-	Y
YEL	314	B+	Y
BLU	315	B-	Y
RED	316	A+	Z
GRN	317	A-	Z
YEL	318	B+	Z
BLU	319	B-	Z

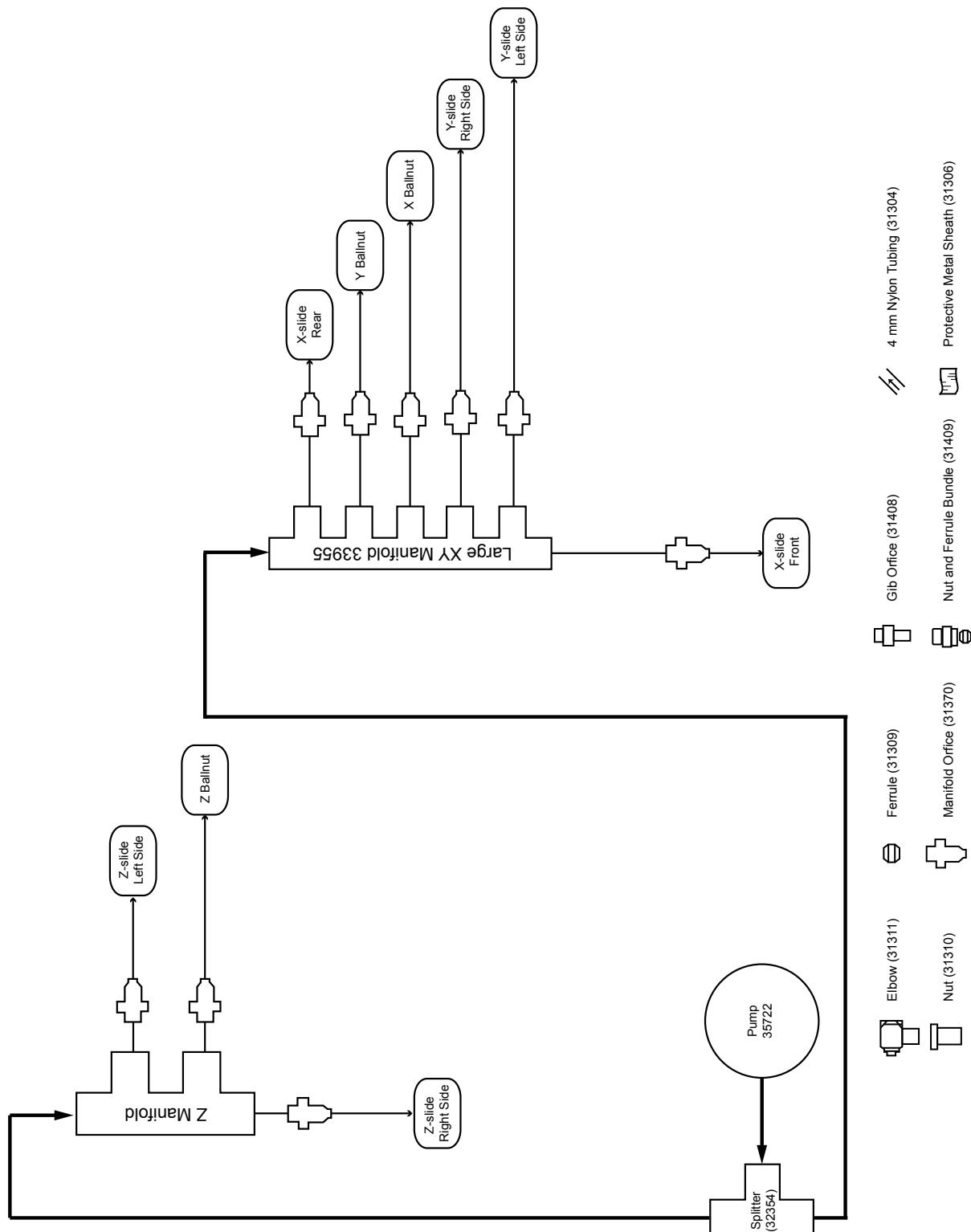
DIP SWITCH SETTINGS FOR X, Y, Z

Diagram illustrating the dip switch settings for the X, Y, and Z axes. The settings are arranged in a grid:

1	off <	on >
2	off <	on >
3	off <	on >
4	off <	on >
5	off <	on >
6	off <	on >

DIAGRAMS AND PARTS LIST

10.7 Lubrication System



PCNC 440 ELECTRICAL SCHEMATIC

