

TORMACH
PCNS
440S



Copyright

Notice

Information is subject to change without notice by Tormach, Inc. For the most recent version of this document, see tormach.com/support.

You're welcome to make copies of this document for evaluating, learning about, and/or using the machine.

You may not charge for any copies you make beyond the cost of printing.

Unless otherwise noted, companies, names, and various data used in examples are fictitious.

To the Reader

We're dedicated to continually improving our documentation and products, and welcome any clarifications, corrections, or suggestions.

Credits

Tormach®, PCNC 440®, Tormach Tooling System (TTS)®, and PathPilot® are trademarks or registered trademarks of Tormach, Inc. Our 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 other patent(s) pending.

Other product or company names may be the trademarks of their respective owners.

Copyright © Tormach, Inc. 2024

IMPORTANT INFORMATION: PLEASE READ FIRST

INSPECT THE ITEMS DELIVERED

Before installing your machine, carefully inspect all delivered items. Complete the following steps upon receiving your shipment to ensure that everything is in proper condition and that any issues are promptly addressed.

1. Inspect the item(s):
 - Photograph any damage that may have occurred during shipping.
 - Note any damage on the delivery receipt before signing for the shipment. If there is extensive visible damage to the machine, refuse to accept the shipment.
 - Verify the received goods against the packing list. If there is any damage or shortages, you must contact Tormach within 30 days of receipt. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.
2. If there are any items that have not yet been delivered, we recommend waiting until you've received all shipments to begin installing the machine. Depending on the product and options ordered, the system may arrive in one or more shipments of:
 - Accessories (if applicable)
 - Machine



Note: The machine system and large accessories are sent by freight carrier. Smaller accessories may be sent by parcel service.

SAVE THESE INSTRUCTIONS!

This document contains important safety warnings and operating instructions for your machine. Before operating this machine in any way, you and all other operators must read and understand all instructions. If you don't, there's a risk of voided warranty, property damage, serious injury, or death. Keep these instructions with your machine so that they're readily accessible.

PURPOSE AND SCOPE OF THIS DOCUMENT

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

GETTING HELP

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

1. Read this document.
2. Read related documents and watch related videos at tormach.com/support.
3. If you still need answers, gather the following information so that we may help you as quickly as possible:
 - Your phone number, address, and company name (if applicable).
 - Machine model and serial number, which are located on the side of the electrical cabinet, next to the Main Disconnect switch.
 - The version of PathPilot that you're running.
 - Any accessories that you have for your machine.
 - A clear and concise description of the issue.
 - Any supporting media and information that you can share with us. For example, you could:
 - Analyze what might have changed since the machine last worked correctly.
 - Record a short video.
 - Take a picture of a part.
 - For software, share log data .zip files, screen captures, or program files.
For information, see "Share Log Data .zip Files" (on the next page).
 - From the PathPilot interface, on the Status tab, record any available information.
 - Use a digital multimeter for voltage readings.

4. Once you've gathered the information in Step 3, contact us in the following ways:
 - a. Create a support ticket: Go to tormach.com/how-to-submit-a-support-ticket
 - b. Phone: (608) 849-8381 (Monday through Friday, 8 a.m. to 5 p.m. U.S. Central Standard Time)

SHARE LOG DATA .ZIP FILES

The controller keeps log data on how the machine has been working, which you can export as a .zip file. This information helps us troubleshoot software situations much faster.

To share log data .zip files:

1. Put a USB drive into the PathPilot controller.
2. From the PathPilot controller, on the **Status** tab, select **Log Data**.
PathPilot creates a file called **logdata_[TODAY'S-DATE].zip**, and saves it on your USB drive.
3. Remove the USB drive from the controller. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.

LIABILITY DISCLAIMER

We've made every effort to provide comprehensive and accurate information, but no warranty or fitness is claimed or implied. All information provided is on an as is basis. The authors, publisher, and Tormach, Inc. ("we", "us", and so on) 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 document.

This document provides guidance on safety precautions and techniques, but because the specifics of any one workshop or other local conditions can vary greatly, we accept no responsibility for machine performance or any damage or injury caused by its use. It's your responsibility to verify that you fully understand the implications of what you're doing and comply with any legislation and codes of practice applicable to your city, state, or nation.

IMPORTANT INFORMATION: PLEASE READ FIRST.....	3
SAFETY.....	19
1.1 Intended Use	20
1.2 Machine Standards	21
1.2.1 American National Safety Institute (ANSI).....	21
1.2.2 Occupational Safety and Health Administration (OSHA).....	21
1.3 Safety Overview.....	22
1.3.1 Safety Messages.....	22
<i>Personal Injury.....</i>	22
<i>Property Damage.....</i>	22
1.4 Machine Safety.....	23
1.4.1 General Shop Safety.....	23
1.4.2 Operational Safety.....	23
<i>General.....</i>	23
<i>Tooling.....</i>	24
<i>Workholding.....</i>	24
1.4.3 Electrical Safety.....	24
1.5 Noise Exposure.....	26
ABOUT YOUR MACHINE.....	27
2.1 Performance Expectations.....	28
2.1.1 Cutting Performance.....	28
2.1.2 Resolution and Accuracy.....	28
2.2 Machine Specifications.....	29
SITE REQUIREMENTS.....	31
3.1 General Site and Space Requirements.....	32
3.1.1 Site Requirements.....	32
3.1.2 Space Requirements.....	32
3.2 Electrical and Power Requirements.....	33
3.2.1 Electrical Requirements.....	33
3.2.2 Power Requirements.....	33
3.2.3 Options for Non-Conforming Sites.....	33
INSTALLATION.....	35
4.1 Before You Begin.....	36
4.1.1 Machine Stand.....	36
4.2 Installation Tools and Items.....	37
4.3 Move the Pallet.....	38

TABLE OF CONTENTS

- 4.4 Assemble the Machine Stand 39
- 4.5 Unpack the Machine Crate 43
- 4.6 Install Core Components 44
 - 4.6.1 Lift and Move the Machine 44
 - 4.6.2 Install the Emergency Stop Box 45
 - 4.6.3 Install Cover Plates 45
 - 4.6.4 Install the Controller 45
 - Install the Controller Arm 45
 - Install the Monitor 46
 - Install the PathPilot Controller 47
 - 4.6.5 Verify the Installation 48
 - Power on the Machine 48
 - Verify Limit Switch Function 49
 - Verify Axes Function 50
 - Verify Spindle Function 50
 - Power off the Machine 51
- 4.7 Install Accessory Components 52
 - 4.7.1 Install the Power Drawbar 52
 - Before You Begin 52
 - Required Tools 52
 - Air Requirements 52
 - Installation 52
 - Prepare the Machine 52
 - Disassemble the Original Power Drawbar 52
 - Assemble the Power Drawbar 53
 - Make Air Connections 55
 - Adjust the Power Drawbar 57
 - Adjust the Drawbar Tension 58
 - About Drawbar Tension 58
 - Adjust the Initial Setup 58
 - 4.7.2 Install the Flood Coolant Kit 59
 - Required Tools 59
 - Set up the Flood Coolant Kit 59
 - 4.7.3 Install the PCNC 440 Automatic Tool Changer 61
 - Required Tools 61
 - Air Requirements 61
 - Before You Begin 62
 - Prepare the Machine 62
 - Disassemble the Accessories 62
 - Disassemble the Coolant Hose 62
 - Disassemble the Enclosure 63
 - Disassemble the Power Drawbar Button 63
 - Mount the Automatic Tool Changer 63
 - Make Air Connections 65
 - (Optional) Install the FRL Filter-Regulator-Lubricator 66

<i>Level the Automatic Tool Changer.</i>	66
<i>Make Electrical Connections.</i>	67
<i>Verify the Installation.</i>	67
<i>Make Final Alignments to the Automatic Tool Changer.</i>	69
Adjust the Tool Tray and Slots.	69
Adjust the Tool Tray Load Position.	69
Adjust the Tilt of the Automatic Tool Changer.	69
Adjust the Tool Tray Rotation.	70
<i>Validate the Automatic Tool Changer Alignment.</i>	71
Setting the Tool Change Height.	71
Adjusting for Rotational Play.	71
Re-Assembly.	71
4.7.4 Install the Enclosure.	71
<i>Before You Begin.</i>	71
<i>Required Tools.</i>	72
<i>Install the Enclosure Panels.</i>	72
Install the Left Rear Panel.	72
Install the Left Side Panel.	72
Install the Left Side Window Retainers.	73
Install the Left Front Panel.	73
Install the Right Side Panel.	73
Install the Right Front Panel.	74
Install the Top Panel.	74
Install the Top and Bottom Trim.	75
<i>Install the Front Doors.</i>	75
Assemble the Front Doors.	75
Install the Front Door Assemblies.	76
<i>Install the Side Windows.</i>	76
4.7.5 Door Switch Installation.	77
<i>Required Tools.</i>	77
<i>Install the Door Switches.</i>	77
<i>3-Hinge Doors (Newer Enclosure).</i>	77
<i>2-Hinge Doors (Older Enclosure).</i>	78
<i>Install the Safety Screws.</i>	80
<i>Make Electrical Connections.</i>	80
<i>Verify the Installation.</i>	81
4.8 Set Up the PathPilot Controller.	82
4.8.1 Specify the Date and Time.	82
4.8.2 Specify the Keyboard Language.	82
4.8.3 Configure the Optional Touch Screen Kit.	82
4.8.4 Update PathPilot.	82
SYSTEM BASICS	83
5.1 System Reference.	84
5.1.1 Machine Table.	84

TABLE OF CONTENTS

5.1.2 Spindle.....	84
<i>About the Spindle.....</i>	84
5.1.3 Axes.....	84
5.2 Basic Controls Reference.....	85
5.2.1 Machine Controls.....	85
5.2.2 PathPilot Interface.....	85
PATHPILOT INTERFACE OVERVIEW.....	87
6.1 About PathPilot.....	88
6.2 Notebook Section.....	89
6.2.1 Main Tab.....	89
6.2.2 File Tab.....	89
6.2.3 Settings Tab.....	89
6.2.4 Offsets Tab.....	90
6.2.5 Conversational Tab.....	90
6.2.6 Probe Tab.....	90
6.2.7 Status Tab.....	90
6.3 Persistent Controls.....	92
6.3.1 Program Control Area.....	92
6.3.2 Position Status Area.....	92
6.3.3 Manual Control Area.....	92
6.4 Keyboard Shortcuts.....	94
6.5 Manage PathPilot Versions.....	95
6.5.1 Download and Install an Update File from the Controller.....	95
6.5.2 Install an Update File from a USB Drive.....	95
6.5.3 Install a Previous Version of an Update File.....	96
PATHPILOT TOOLS AND FEATURES.....	97
7.1 Create and Load G-Code Files.....	98
7.1.1 Load G-Code.....	98
<i>Transfer Files to and From the Controller.....</i>	98
About System Files.....	98
<i>Preview G-Code Files.....</i>	98
<i>Access Recent G-Code Files.....</i>	99
<i>Close the Current Program.....</i>	99
7.1.2 Edit G-Code.....	99
<i>Edit G-Code with a Text Editor.....</i>	99
<i>Edit G-Code with Conversational Programming.....</i>	99
Change the Step Order.....	100
Create a New Job Assignment.....	100
Load an Existing G-Code File.....	100
Edit a Job Assignment.....	100
7.1.3 Read G-Code.....	100
<i>Expand the G-Code Tab.....</i>	100
About the G-Code Tab.....	100

<i>Search in the Code.</i>	100
<i>Set a New Start Line.</i>	101
<i>Lead-In Moves.</i>	101
<i>Change the View of the Tool Path Display.</i>	102
<i>About the Tool Path Display.</i>	102
7.1.4 Use Conversational Programming.	103
<i>About Conversational Programming.</i>	103
<i>Create a Face on a Part.</i>	103
<i>About Facing.</i>	104
<i>Facing in PathPilot.</i>	104
Facing Reference.	104
<i>Create a Profile on a Part.</i>	104
<i>About Profiling.</i>	105
<i>Profiling in PathPilot.</i>	105
Profiling Reference.	106
<i>Create a Pocket on a Part.</i>	106
<i>About Pockets.</i>	107
<i>Creating Pockets in PathPilot.</i>	107
<i>About Circular Pockets.</i>	107
<i>About Rectangular Pockets.</i>	107
<i>Create Hole Locations on a Part (Drill).</i>	108
<i>About Drilling.</i>	109
<i>Drilling in PathPilot.</i>	109
Create a Drilling Sequence.	109
Drilling Reference.	109
<i>Create Threads on a Part.</i>	110
<i>About Thread Milling.</i>	111
<i>Thread Milling in PathPilot.</i>	111
Thread Milling Reference.	111
<i>Create Text to Engrave on a Part.</i>	111
<i>About Engraving.</i>	112
<i>Import a DXF File.</i>	112
Working with Layers and Shapes.	113
<i>Change the Layer or Shape Cut Order.</i>	113
<i>Adjust the Tree View Window.</i>	113
Working in the Preview Window.	113
7.2 Machine Settings and Accessories.	114
7.2.1 Enable an Internet Connection.	114
<i>Connect to Dropbox.</i>	114
<i>Enable Automatic Updates.</i>	115
7.2.2 Change the Network Name.	116
7.2.3 Change the Screen Orientation.	116
<i>About Portrait Screen Layout.</i>	116
7.2.4 Specify the Tool Change Method.	117
<i>About Automatic Tool Changes.</i>	118

TABLE OF CONTENTS

<i>About Manual Tool Changes</i>	118
7.2.5 Enable the On-Screen Keyboard.....	118
<i>About Soft Keyboards</i>	118
7.2.6 Enable Tooltips.....	118
7.2.7 Enable the USB M-Code I/O Interface Kit.....	119
7.2.8 Enable Feeds and Speeds Suggestions in Conversational Routines.....	119
7.2.9 Use a USB Camera.....	119
<i>About USB Cameras</i>	119
<i>Manual Recording</i>	120
<i>Automatic E-Stop Loop Recording ("Dashcam")</i>	120
<i>Review Video and Image Files</i>	121
File Naming Convention.....	121
<i>G-Code Commands</i>	121
File Naming Conventions.....	121
Use M01 to Take Pictures.....	121
7.3 Set Up G-Code Programs	123
7.3.1 Use a Probe with PathPilot.....	123
<i>Set Up the Probe</i>	123
<i>Use a Probe to Find a Feature's Location</i>	123
<i>Use a Probe to Set Work Offset Zeroes</i>	123
Set the X and Y Work Offset Zero on the Corner of a Feature.....	123
Set the Work Offset Zeroes on a Feature.....	123
<i>Use a Probe to Find the Center of a Feature</i>	124
Find the Center of a Pocket.....	124
Find the Center of a Slot.....	124
Find the Center of a Rectangular Boss.....	124
Find the Center of a Circular Boss.....	124
Find the Center Rotation of an A-Axis.....	125
7.3.2 Set Tool Length Offsets.....	125
<i>About Tool Offsets</i>	125
<i>Touch Off the Tool Length Offsets</i>	126
Set a Known Reference Height.....	126
Measure Tools Using a Known Reference Height.....	126
<i>Use a Height Gauge to Measure Tools</i>	127
7.3.3 Set Work Offsets.....	127
<i>About Work Offsets</i>	127
7.3.4 View Work Offsets.....	128
7.3.5 View Available G-Code Modes.....	128
7.4 Run G-Code Programs	129
7.4.1 Bring the Machine Out of Reset.....	129
<i>About Reset Mode</i>	129
7.4.2 Disable Limit Switches.....	129
7.4.3 View the Active Axis to Jog.....	129
7.4.4 Jog the Machine.....	129
<i>Jog in Continuous Velocity Mode</i>	130

<i>Jog in Step Mode</i>	130
7.4.5 View the Current Machine Position.....	130
7.4.6 Reference the Machine.....	131
<i>About Referencing</i>	131
7.4.7 Start a Program.....	131
<i>About Cycle Start</i>	131
<i>Cycle Start Reference</i>	131
7.4.8 Stop Machine Motion.....	131
7.4.9 Operate the Coolant Pump.....	131
<i>About Coolant</i>	132
7.4.10 View the Active G-Code Modes.....	132
7.4.11 View the Distance to Go.....	132
<i>About Distance to Go</i>	132
7.5 Control G-Code Programs	133
7.5.1 Use the Feed Hold Function.....	133
<i>About Feed Hold</i>	133
7.5.2 Use the Feed Rate Override Function.....	133
<i>About Feed Rate Override</i>	133
7.5.3 Use M01 Break Mode.....	134
<i>About M01 Break</i>	134
<i>Display Information and Capture Images During an M00 or M01 Break</i>	134
Display Information with Images.....	134
Display Information with Text.....	134
Capture Images with a USB Camera.....	134
7.5.4 Use the Maxvel Override Function.....	134
<i>About Maxvel Override</i>	135
7.5.5 Use Single Block Mode.....	135
<i>About Single Block</i>	135
7.5.6 Use the Spindle Override Function.....	135
<i>About Spindle Override</i>	135
7.5.7 Change the Feed Rate.....	136
<i>About Feed Rates</i>	136
7.5.8 Change the Spindle Speed.....	136
<i>About Spindle Controls</i>	136
<i>Spindle Controls Reference</i>	136
7.5.9 Change the Tool Number.....	136
<i>About M6 G43</i>	136
7.5.10 Use a G30 Position.....	137
<i>About G30</i>	137
7.5.11 View the Tool Length.....	137
7.5.12 Manually Enter Commands.....	137
<i>About the MDI Line DRO Field</i>	138
<i>Admin Commands Reference</i>	138
7.5.13 Copy Recently Entered Commands.....	139
7.5.14 Limit G30 Moves.....	139

TABLE OF CONTENTS

<i>About G30.</i>	139
7.5.15 Use Feeds and Speeds Suggestions.	139
<i>Refresh DRO Field Values.</i>	140
<i>Use Additional Provided Information.</i>	140
<i>Enable Feeds and Speeds Suggestions in Conversational Routines.</i>	140
<i>Create Tool Descriptions.</i>	140
Manually Enter Tool Descriptions.	141
Automatically Generate Tool Descriptions.	141
<i>Tool Keywords Reference.</i>	141
7.6 System File Management	143
7.6.1 Manage System Files.	143
<i>About System Files.</i>	143
7.6.2 Create Backup Files.	143
<i>About Backup Files.</i>	144
7.6.3 Restore Backup Files.	144
7.6.4 Import and Export the Tool Table.	145
<i>Import a .csv File.</i>	145
<i>Export the Tool Table as a .csv File.</i>	145
BASIC OPERATIONS.	147
8.1 Start the Machine	148
8.2 Reference the Machine	149
8.2.1 About Referencing.	149
8.3 Jog the Machine	150
8.3.1 About Jogging.	150
<i>About Continuous Velocity Jogging.</i>	150
<i>About Step Jogging.</i>	150
8.3.2 Jog Controls Reference.	151
<i>Jogging in PathPilot.</i>	151
<i>Jogging with the Keyboard.</i>	151
<i>Jogging with the (Optional) Jog Shuttle.</i>	151
8.4 Manually Control the Spindle	152
8.4.1 Change the Spindle Speed Range.	152
8.4.2 About the Spindle.	152
8.4.3 Spindle Controls Reference.	152
8.5 Load G-Code	153
8.6 Set Up Tooling	154
8.6.1 Install a Tool in a Set Screw Tool Holder.	154
8.6.2 Install a Tool in an ER Collet Tool Holder.	154
8.6.3 Install a Drill Chuck in a Jacobs Taper Arbor.	154
8.7 Set Tool Length Offsets	155
8.7.1 About Tool Offsets.	155
8.7.2 Touch Off the Tool Length Offsets.	155
<i>Set a Known Reference Height.</i>	155
<i>Measure Tools Using a Known Reference Height.</i>	156

8.7.3 Use a Height Gauge to Measure Tools.....	157
8.8 Set Work Offsets.....	158
8.8.1 About Work Offsets.....	158
8.9 Operate the Coolant Pump.....	159
8.9.1 About Coolant.....	159
FIRST PART TUTORIAL.....	161
9.1 About This Tutorial.....	162
9.2 Before You Begin.....	163
9.3 Required Items and Tools.....	164
9.4 Set Up Tooling.....	165
9.4.1 Install a Tool in a Set Screw Tool Holder.....	165
9.4.2 Install a Tool in an ER Collet Tool Holder.....	165
9.5 Set Up the Machine.....	166
9.5.1 About Referencing.....	166
9.6 Mount the Workpiece.....	167
9.7 Set the Work Offsets.....	168
9.7.1 Set the Z Zero.....	168
9.7.2 Set the X/Y Zeroes.....	168
9.7.3 About Work Offsets.....	168
9.7.4 About True Positive Tool Length.....	169
9.7.5 About X/Y Part Zeroes.....	169
9.8 Set Tool Offsets.....	170
9.8.1 Touch Off the Tool Offsets.....	170
9.8.2 Measure Tools Using a Known Reference Height.....	170
9.9 Complete the Milling Operations.....	171
9.9.1 Create a Pocket.....	171
<i>Load the Tool.....</i>	<i>171</i>
<i>Write the G-Code with Conversational.....</i>	<i>171</i>
<i>Run the Program.....</i>	<i>172</i>
9.9.2 Engrave the Text.....	172
<i>Write the G-Code with Conversational.....</i>	<i>172</i>
<i>Run the Program.....</i>	<i>173</i>
PROGRAMMING.....	175
10.1 Programming Overview.....	176
10.1.1 About G-Code Programming Language.....	176
10.1.2 G-Code Formatting Reference.....	176
<i>Line Numbers.....</i>	<i>176</i>
<i>Words.....</i>	<i>176</i>
<i>Letters.....</i>	<i>177</i>
<i>Values.....</i>	<i>177</i>
<i>Order of Execution.....</i>	<i>178</i>
<i>Modal Groups.....</i>	<i>179</i>
<i>Comments.....</i>	<i>180</i>

TABLE OF CONTENTS

10.1.3 Supported G-Codes Reference.....	180
10.2 Programming G-Code.....	182
10.2.1 About the Examples Used.....	182
10.2.2 Rapid Linear Motion (G00).....	182
10.2.3 Linear Motion at Feed Rate (G01).....	182
10.2.4 Arc at Feed Rate (G02 and G03).....	183
<i>Radius Format Arc.....</i>	<i>183</i>
<i>Center Format Arc.....</i>	<i>183</i>
Arc in XY Plane.....	184
Arc in XZ Plane.....	184
Arc in YZ Plane.....	184
10.2.5 Dwell (G04).....	185
10.2.6 Set Offsets (G10).....	185
<i>Set Tool Table (G10 L1).....</i>	<i>185</i>
<i>Set Coordinate System (G10 L2).....</i>	<i>185</i>
<i>Set Tool Table (G10 L10).....</i>	<i>185</i>
<i>Set Tool Table (G10 L11).....</i>	<i>186</i>
<i>Set Coordinate System (G10 L20).....</i>	<i>186</i>
10.2.7 Plane Selection (G17, G18, G19).....	186
10.2.8 Length Units (G20 and G21).....	186
10.2.9 Return to Predefined Position (G28 and G28.1).....	186
10.2.10 Return to Predefined Position (G30 and G30.1).....	187
10.2.11 Automatically Measure Tool Lengths with an ETS (G37 and G37.1).....	187
<i>Move to G37 Position Over ETS (G37.1).....</i>	<i>187</i>
<i>Move and Measure Tool Length (G37).....</i>	<i>187</i>
10.2.12 Straight Probe (G38.x).....	188
<i>Use the Straight Probe Command.....</i>	<i>188</i>
10.2.13 Cutter Compensation (G40, G41, G42).....	189
10.2.14 Dynamic Cutter Compensation (G41.1 and G42.1).....	189
10.2.15 Apply Tool Length Offset (G43).....	189
10.2.16 Engrave Sequential Serial Number (G47).....	190
10.2.17 Cancel Tool Length Compensation (G49).....	190
10.2.18 Absolute Coordinates (G53).....	190
10.2.19 Select Work Offset Coordinate System (G54 to G54.1 P500).....	190
10.2.20 Set Exact Path Control Mode (G61).....	191
10.2.21 Set Blended Path Control Mode (G64).....	191
10.2.22 Distance Mode (G90 and G91).....	191
10.2.23 Arc Distance Mode (G90.1 and G91.1).....	191
10.2.24 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	191
10.2.25 Feed Rate Mode (G93, G94, and G95).....	192
10.2.26 Spindle Control Mode (G96 and G97).....	192
10.3 Programming Canned Cycles.....	194
10.3.1 Canned Cycles Reference.....	194
<i>Supported Canned Cycles.....</i>	<i>194</i>
10.3.2 High Speed Peck Drill (G73).....	195

10.3.3 Cancel Canned Cycles (G80).....	195
10.3.4 Drilling Cycle (G81).....	195
10.3.5 Simple Drilling Cycle (G82).....	197
10.3.6 Peck Drilling Cycle (G83).....	197
10.3.7 Boring Cycle (G85).....	197
10.3.8 Boring Cycle (G86).....	197
10.3.9 Boring Cycle (G88).....	197
10.3.10 Boring Cycle (G89).....	197
10.4 Programming M-Code.....	199
10.4.1 Supported M-Codes Reference.....	199
10.4.2 Program Stop and Program End (M00, M01, M02, and M30).....	199
<i>Display Information and Capture Images During an M00 or M01 Break.....</i>	<i>199</i>
Display Information with Images.....	199
Display Information with Text.....	200
Capture Images with a USB Camera.....	200
10.4.3 Spindle Control (M03 and M05).....	200
10.4.4 Tool Change (M06).....	200
10.4.5 Coolant Control (M07, M08, and M09).....	201
10.4.6 Override Control (M48 and M49).....	201
10.4.7 Feed Override Control (M50).....	201
10.4.8 Spindle Speed Override Control (M51).....	201
10.4.9 Set Current Tool Number (M61).....	201
10.4.10 Set Output State (M64 and M65).....	201
10.4.11 Wait on Input (M66).....	201
10.5 Programming Input Codes.....	203
10.5.1 Feed Rate (F).....	203
10.5.2 Spindle Speed (S).....	203
10.5.3 Change Tool Number (T).....	203
10.6 Advanced Programming.....	204
10.6.1 Parameters.....	204
<i>Parameters Reference.....</i>	<i>204</i>
Parameter Syntax.....	204
Parameter Scope.....	204
Behavior of Uninitialized Parameters.....	204
Parameter Mode.....	204
Persistence and Volatility.....	204
Intended Use.....	204
<i>Numbered Parameters Reference.....</i>	<i>205</i>
Read-Only Parameters.....	205
<i>Subroutine Parameters Reference.....</i>	<i>205</i>
<i>Named Parameters Reference.....</i>	<i>205</i>
10.6.2 Expressions.....	206
<i>Binary Operators Reference.....</i>	<i>206</i>
<i>Functions Reference.....</i>	<i>206</i>
10.6.3 Subroutines.....	207

TABLE OF CONTENTS

<i>Subroutines Reference</i>	207
<i>Conditional Subroutines Reference</i>	208
if/endif.....	208
if/elseif/else/endif.....	208
<i>Repeating Subroutines Reference</i>	208
<i>Looping Subroutines Reference</i>	208
do/while.....	209
while/endwhile.....	209
MACHINE MAINTENANCE	211
11.1 Maintenance Safety	212
11.1.1 All Maintenance Procedures.....	212
11.1.2 Swarf Maintenance Procedures.....	212
11.2 Maintenance Schedules	213
11.2.1 Daily.....	213
11.2.2 Weekly.....	213
11.2.3 Monthly.....	213
11.2.4 Semi-Annually.....	213
11.3 Regular Maintenance	214
11.3.1 Clean the Way Covers.....	214
11.3.2 Clean the Coolant System.....	214
<i>About Cutting Fluid</i>	214
<i>Cutting Fluid Reference</i>	214
11.3.3 Examine the Drawbar.....	215
11.3.4 Examine the Enclosure Windows.....	215
11.3.5 Lubricate the Machine.....	215
<i>About the Manual Oiler</i>	215
<i>Lubrication System Reference</i>	216
11.3.6 Examine the Spindle Belt.....	216
11.3.7 Examine the Tormach Tooling System (TTS) Collet.....	216
11.3.8 Prevent Rust.....	216
<i>Remove Rust</i>	216
11.4 Advanced Maintenance Adjustments	217
11.4.1 About Advanced Maintenance.....	217
11.4.2 Definitions.....	217
<i>Lost Motion</i>	217
<i>Backlash</i>	217
11.4.3 How to Measure Lost Motion.....	217
<i>Required Tools</i>	218
<i>To Measure X-Axis Backlash</i>	218
11.4.4 Adjust the Gibs.....	218
<i>About Gibs</i>	219
11.4.5 Adjust the Angular Contact Bearing Pre-Load.....	220
<i>Required Tools</i>	220
<i>To Adjust the Angular Contact Bearing Pre-Load</i>	220

About Angular Contact Bearing Pre-Load220

 11.4.6 Adjust the Geometry of Precision Mating Surfaces..... 220

 11.4.7 Adjust the Spindle Bearings..... 221

11.5 Transporting the Machine..... 222

DIAGRAMS AND PARTS LISTS..... 223

12.1 Spindle Head 224

12.2 Z-Axis Column 227

12.3 X-Y Axis 229

12.4 Electrical Cabinet..... 232

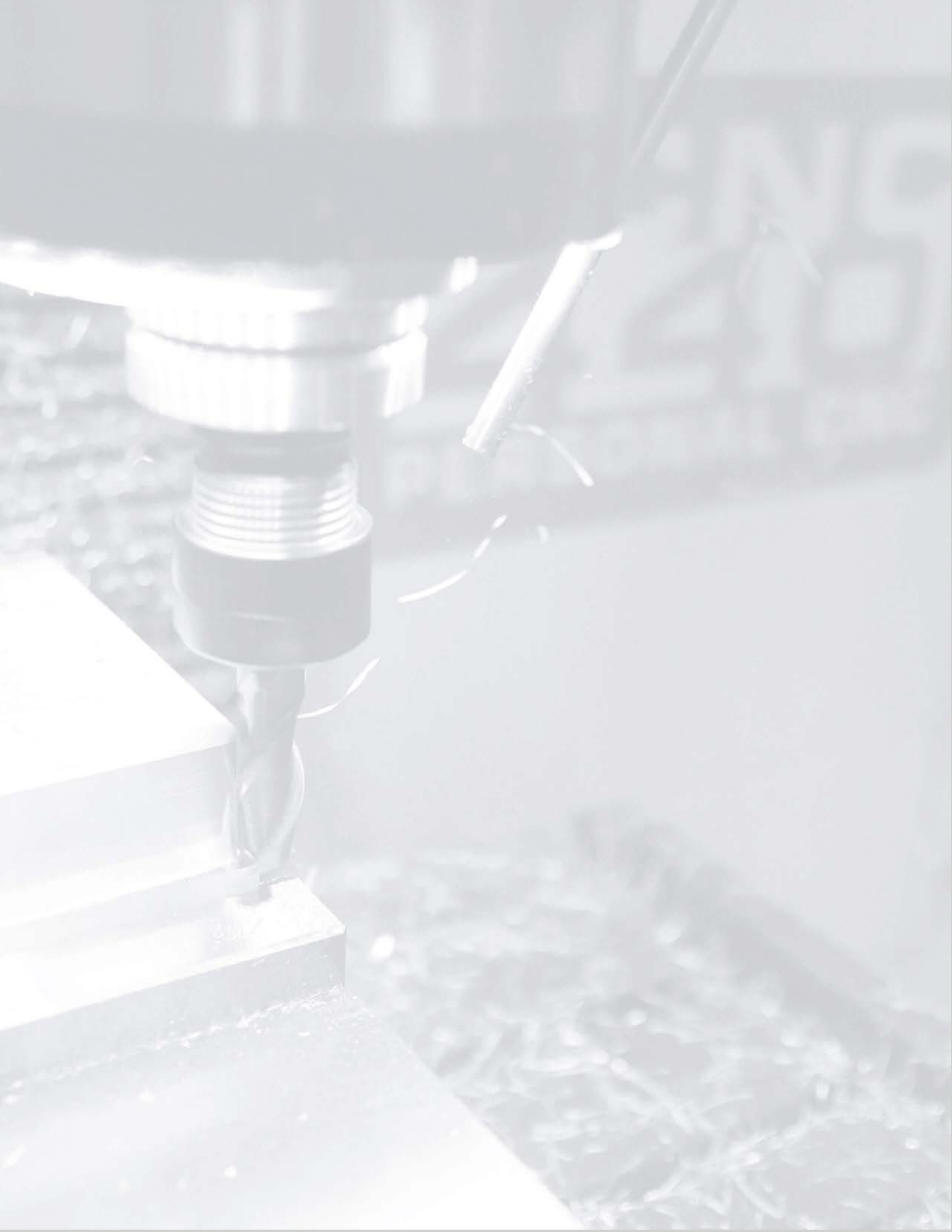
12.5 Connections.....234

12.6 Stepper Connections..... 235

12.7 Lubrication/Oil System Diagram.....236

ELECTRICAL SCHEMATICS..... 237

13.1 Electrical Schematic238



SAFETY

IN THIS SECTION, YOU'LL LEARN:

- About the standards and safety precautions associated with this machine.



Before operating the machine in any way, you must read and understand this section.

Safe operation of the machine depends on its proper use and the precautions you take. Only trained personnel — with a clear and thorough understanding of its operation and safety requirements — shall operate this machine.

CONTENTS

1.1 Intended Use	20
1.2 Machine Standards	21
1.3 Safety Overview	22
1.4 Machine Safety	23
1.5 Noise Exposure	26

1: SAFETY

1.1 Intended Use

1.1 INTENDED USE

This machine is intended for general-purpose, computer numerical control (CNC) machining in the following applications:

- Educational environments
- Hobby applications
- Light production
- Prototyping
- Research and development
- Secondary operations

The intended use includes:

- Appropriate workholding, toolholding, tooling, coolant systems, and machining parameters.
- Machining of conventional, non-abrasive materials such as ferrous and non-ferrous metals below 60 HRC, woods, and plastics.

The intended use **does not** include machining materials that:

- Are abrasive, carcinogenic, explosive, flammable, radioactive, or toxic
- Produce aerosols or fine particulates when machined

The intended use **does not** include the following materials (not a full list):

- Beryllium and its alloys
- Ceramics
- Fiberglass
- G10 fiberglass laminate
- Graphite
- Magnesium and its alloys

To safely operate products, you must obey all safety precautions and warnings that are on the machines and in the documentation.

1.2 MACHINE STANDARDS

When installed and operated as intended (see "Intended Use" (on the previous page)), this machine complies with the following standards. You must follow the requirements listed in the standards so that the machine remains compliant.

1.2.1 American National Safety Institute (ANSI)

- **ANSI B11.TR3-2000** *Risk Assessment and Risk Reduction — A Guideline to Estimate, Evaluate, and Reduce Risks Associated with Machine Tools*
- **ANSI B11.8-2001** *Safety Requirements for Manual Milling, Drilling, and Boring Machines with or without Automatic Control*
- **ANSI B11.19-2010** *Performance Requirements for Safeguarding*
- **ANSI B11.23-2002** *Safety Requirements for Machining Centers and Automatic Numerically Controlled Milling, Drilling, and Boring Machines*

1.2.2 Occupational Safety and Health Administration (OSHA)

- **OSHA 1910.212** *General Requirements for All Machines*
- **OSHA 3067** *Concepts and Techniques of Machine Safeguarding*

1: SAFETY

1.3 Safety Overview

1.3 SAFETY OVERVIEW

Any machine tool is potentially dangerous. A CNC machine's automation presents added risk not present in a manual machine.

Before operating the machine in any way, you must read and understand this section.

- Read and understand all safety messages used in this document.
- Locate and understand all safety decals on the machine.
- Locate and become familiar with all information decals on the machine.

1.3.1 Safety Messages

The following examples show the standard safety message types used to draw your attention to important information. The standards distinguish between personal injury safety messages and property damage warning messages.

Personal Injury

Personal injury safety messages have safety alert symbols and the following hazard level labels:



DANGER! Indicates a hazard with a high level of risk which, if not avoided, will result in death or serious injury.



WARNING! Indicates a hazard with a medium level of risk which, if not avoided, can result in death or serious injury.



CAUTION! Indicates a hazard with a low level of risk which, if not avoided, can result in minor or moderate injury.

Property Damage

NOTICE! Indicates a hazard which, if not avoided, can cause property damage.

1.4 MACHINE SAFETY



Before operating the machine in any way, you must read and understand this section.

Safe operation of the machine depends on its proper use and the precautions you take. Only trained personnel — with a clear and thorough understanding of its operation and safety requirements — shall operate this machine.

1.4.1 General Shop Safety	23
1.4.2 Operational Safety	23
1.4.3 Electrical Safety	24

1.4.1 General Shop Safety

- ✓ Verify that only qualified machinery maintenance professionals install, set up, or perform maintenance on this machine.
- ✓ Keep the work area well-lit. Use additional lighting if needed. The work area should be illuminated to a minimum of 500 lx.
- ✓ Keep the work area temperature- and humidity-controlled.
- ✓ Remove loose-fitting clothing, neckties, gloves, and jewelry.
- ✓ Tie up long hair and secure it under a hat.
- ✓ Wear safety eye protection rated for ANSI Z87+.
- ✓ Wear closed-toed safety shoes.
- ✓ Wear ear protection when you expect the machine or the machining processes to exceed safe exposure limits. For information, see "Noise Exposure" (page 26).
- ✓ Keep the work area clean and free of clutter. Machine motion can occur if controls are accidentally activated.
- ✓ Immediately clean up spills after they occur.
- ✗ Never operate the machine after consuming alcohol or taking medication that could prevent you from safely operating the machine.
- ✗ Never operate the machine while tired or otherwise impaired.
- ✗ Never operate the machine in an explosive (ATEX) atmosphere. Such explosive atmospheres include explosive gases, vapors, mists, powders, and dusts.

1.4.2 Operational Safety

General

- ✓ Understand that the machine is automatically controlled and can start at any time.
- ✓ If anyone enters the machining envelope during operation, immediately push in the Emergency Stop button. In case of a power loss, turn the Door Switch Override to the Unlocked position and open the enclosure doors.
- ✓ Become familiar with all physical and software controls.
- ✓ Always use a chip scraper or brush when clearing away chips, oil, or coolant.
- ✓ Examine all tools, fixtures, workpieces, and guarding for signs of damage. Replace any damaged components as soon as you find them.
The enclosure and other guards may not stop all types of projectiles, like broken tools or loose workpieces.
- ✓ Stop the machine and verify that all machine motion has completely stopped before doing any of the following:
 - ✓ Adjusting a part, fixture, or coolant nozzle.
 - ✓ Changing belt or pulley positions.
 - ✓ Changing tools or parts.
 - ✓ Clearing away chips, oil, or coolant.
 - ✓ Reaching into any part of the machine's motion envelope.
 - ✓ Removing protective shields or safeguards.
 - ✓ Taking measurements.
 - ✓ Doing any other action inside the machine enclosure.
- ✓ Use flood or MQL (mist) coolant as required by the machining operation.
- ✓ Only use coolants designed for metal working applications such as soluble oils, semi-synthetic, or synthetic coolants.
- ✓ Read the Safety Data Sheet (SDS) for all workpiece materials, coatings, coolants (flood or MQL), lubricants, and other consumables. Chips, dust, and vapors from certain materials can be toxic or otherwise harmful.
- ✓ Dispose of scrap and swarf according to local regulations and guidelines.
- ✓ Thoroughly read all safety precautions and instructions.

1: SAFETY

1.4 Machine Safety

- ✓ When machining an unproven program, use feed, speed, and maximum velocity overrides, Distance-to-Go (DTG) displays, single block, feed hold, and other control features.
- ✓ Follow all appropriate "Machine Standards" (page 21).
- ✗ Never enter the machining envelope.
- ✗ Never reach around a guard.
- ✗ Never allow the machine to run unattended.
- ✗ Never obstruct the Emergency Stop button or any other controls.
- ✗ Never allow untrained operators to install, operate, or maintain the machine.
- ✗ Never modify, defeat, or bypass safety devices or interlocks.
- ✗ Never machine abrasive, carcinogenic, explosive, flammable, radioactive, or toxic materials. Such materials include, but are not limited to:
 - ✗ Beryllium and its alloys
 - ✗ Ceramic
 - ✗ Fiberglass
 - ✗ G10 fiberglass laminate
 - ✗ Graphite
 - ✗ Lead and its alloys
 - ✗ Magnesium and its alloys
- ✗ Never allow swarf to accumulate on or within the machine.
- ✗ Never use flammable liquids (like alcohol, diesel fuel, or kerosene) in the machine's coolant system.
- ✗ Never use water, coolants without rust inhibitors, or straight cutting oil in the machine's coolant system.

Tooling

- ✓ Use appropriate speeds, feeds, and cutting parameters for your machine, machine operation, material, and tooling.
- ✓ Use tools and tool holders that are suitable for the current operation.
- ✓ Examine tools for signs of damage. Replace any damaged tools as soon as you find them.
The enclosure and other guards may not stop all types of projectiles, like broken tooling.

- ✗ Never use unbalanced tooling or spindle fixtures.
- ✗ Never use tools that are larger or longer than necessary.
- ✗ Never use tools at speeds above their operational limits.

Workholding

- ✓ Secure workpieces with appropriate workholding devices (like a milling vise or strap clamps).
- ✓ Verify that the workpiece is adequately secured.
- ✓ Position clamps and workholding devices clear of any tool paths.
- ✓ Remove cutoff workpieces and other large chips before starting the machine.
- ✗ Never leave tools, stock, or other loose items inside the machine.

1.4.3 Electrical Safety



WARNING! Electrical Shock Hazard: You must power off the machine before making any electrical connections. If you don't, there's a risk of electrocution or shock.

- ✓ Power off the machine before servicing.
- ✓ Use an approved lockout/tagout system to secure the machine's Main Disconnect in the **OFF** position before servicing the machine.
- ✓ Understand that certain electrical components can retain dangerous electrical voltages, even after the machine is powered off and all power is removed from the system.
- ✓ Understand that certain installation, maintenance, and troubleshooting procedures — for the machine and certain accessories — require access to or modification of wiring inside of the electrical cabinet. Only qualified electrical machinery technicians shall perform these procedures.
- ✓ Confirm that the mains voltage conforms to requirements before connecting the machine.
For more information, see "Electrical and Power Requirements" (page 33).
- ✓ Confirm that the machine installation meets all codes and regulations of your locality.
- ✓ Confirm that electrical connections are performed by a certified electrician.

- ✔ Lock the electrical cabinet door and remove the keys when the machine is not being serviced to prevent unqualified or unauthorized personnel from accessing the electrical cabinet.
- ✘ Never operate the machine with the electrical cabinet door open.
- ✘ Never reach into the electrical cabinet with the machine powered on.
- ✘ Never modify the machine's electronics.
- ✘ Never drill into the electrical cabinet.

1: SAFETY

1.5 Noise Exposure

1.5 NOISE EXPOSURE



CAUTION! Noise Exposure Hazard: If you expect the machine or the machining processes to exceed safe exposure limits, you must wear ear protection.

With the axes moving and the spindle rotating (but not cutting), the emission sound pressure level at the operator control station is 70 dB(A).

Emission sound pressure levels vary depending on the machining processes. Factors that influence the actual level of noise exposure may include:

- Characteristics of the work room
- Other sources of noise (like the number of machines, or other adjacent processes)

The permissible noise exposure level may vary between countries. You must evaluate of the hazard and risk based on your locality, and mitigate as necessary.

ABOUT YOUR MACHINE

IN THIS SECTION, YOU'LL LEARN:

- About this machine's specifications.

CONTENTS

2.1 Performance Expectations.....	28
2.2 Machine Specifications.....	29

2: ABOUT YOUR MACHINE

2.1 Performance Expectations

2.1 PERFORMANCE EXPECTATIONS

Each machine has a Certificate of Inspection, which details quality assurance measurements performed by members of our quality assurance team prior to shipping.

2.1.1 Cutting Performance

This machine is capable of cutting a wide variety of materials (for information, see "Intended Use" (page 20)) at or near their recommended feeds and speeds. Make sure that your workpiece is held as rigidly as possible and use the most rigid tooling available for roughing cuts. Verify that the programmed operations do not exceed the available spindle power.

- **Spindle Speed Range**
Two speed ranges are available:
 - **Low** 300 rpm to 4500 rpm
 - **High** 650 rpm to 10,000 rpm
- **Spindle Power Rating** 0.75 hp (0.56 kW)
- **Maximum Feed Rate**
 - **X- and Y-Axis** 135 IPM (3.43 m/min)
 - **Z-Axis** 110 IPM (2.7 m/min)

2.1.2 Resolution and Accuracy

Accuracy is heavily influenced by the techniques that the machinist uses. A skilled machinist can deliver accuracy that exceeds the specified accuracy from the manufacturer; an inexperienced machinist may have difficulty delivering the specified accuracy. We can't predict operator accuracy, but the specified accuracy is an important reference point.

- **Resolution** 0.0001"



Note: The resolution of motion is the minimum discrete positional move.

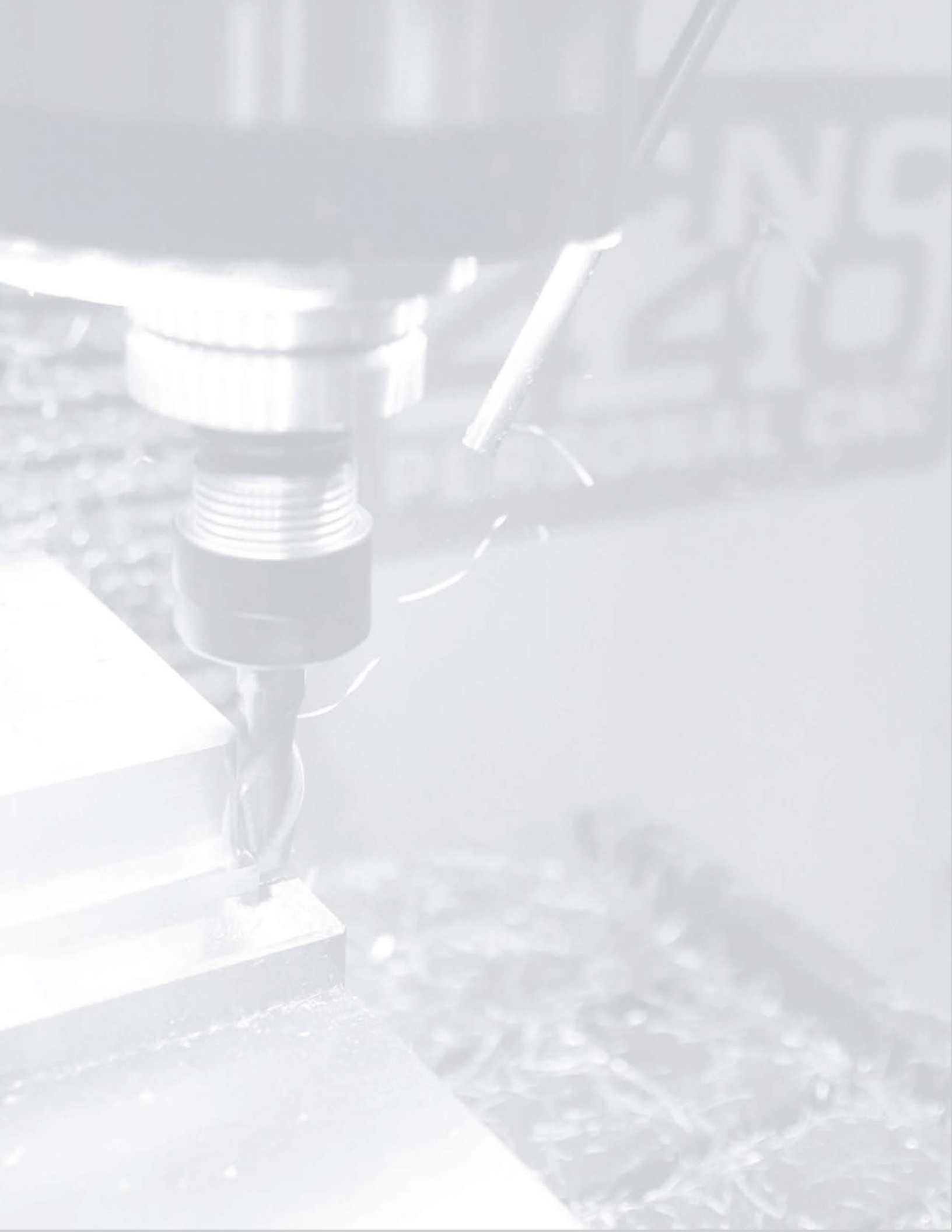
- **Positional Accuracy** ≤ 0.0016 in./10 in.



Note: The positional accuracy includes additional contributing factors (like compressibility of bearings, ball screw windup, and friction).

2.2 MACHINE SPECIFICATIONS

Travels	
X-Axis	10" (254 mm)
Y-Axis	6.25" (159 mm)
Z-Axis	10" (254 mm)
Spindle-to-Table Maximum Clearance	12" (305 mm)
Spindle Centerline to Machine Column	6.5" (165 mm)
Spindle	
Spindle Power	0.75 hp (0.56 kW)
Spindle Maximum Speed	10,000 rpm
Transmission	Poly-V Belt
Spindle Taper	R8
Thread Machining	Thread Mill
Maximum Feed Rate	
X- and Y-Axis	135 IPM (3.43 m/min)
Z-Axis	110 IPM (2.7 m/min)
Axis Motor	Stepper Driven
Power	
Primary Power Required	Single-Phase 115 Vac, 50/60 Hz
Recommended Circuit Amperage	Dedicated 15 A breaker
Table	
Table Size	18" × 6.3" (457 mm × 160 mm)
T-Slot Size	3/8" (9.5 mm) (three slots)
Maximum Weight on Table	150 lb (68 kg)
Overall Dimensions	
Machine Size	42" × 36" (1.1 m × 0.9 m)
Overall System Height	72" (1.8 m)
Typical System Weight	600 lb (272 kg)



SITE REQUIREMENTS

IN THIS SECTION, YOU'LL LEARN:

- About the site requirements of this machine (including electrical and power requirements).



Before operating the machine in any way, you must read and understand this section.

CONTENTS

3.1 General Site and Space Requirements	32
3.2 Electrical and Power Requirements	33

3: SITE REQUIREMENTS

3.1 General Site and Space Requirements

3.1 GENERAL SITE AND SPACE REQUIREMENTS

When choosing a location for your machine, you must verify that it meets all requirements outlined in this section.

3.1.1 Site Requirements

You must verify that the area:

- Allows for unrestricted access to machine controls.
- Conforms to the following:
 - **Primary Power Required** Single-Phase 115 Vac, 50/60 Hz
 - **Recommended Circuit Amperage** Dedicated 15 A breaker



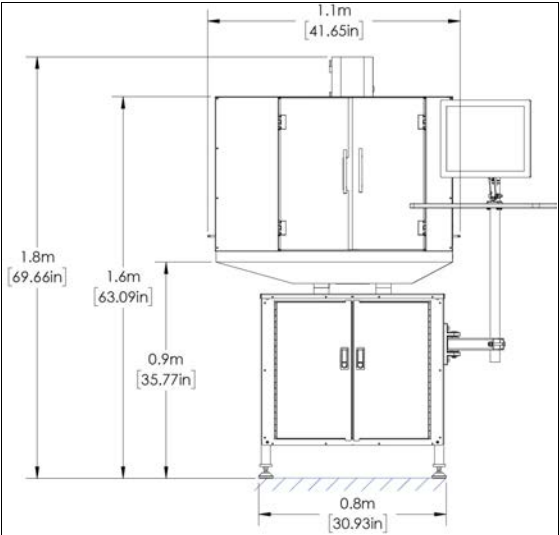
Note: For more information, see "Electrical and Power Requirements" (on the next page).

- Has clean, dry, compressed air.
- Has one continuous slab sufficient to support the weight of the machine, accessories, and any additional equipment.
- Is a dry, properly ventilated, and well-lit internal space that conforms to the following temperature and humidity requirements:
 - **Operating Temperature Range** 40°F-100°F (5°C-38°C)
 - **Humidity Range** 5%-95% (non-condensing)
- Provides for unobstructed machine motion and operation.

3.1.2 Space Requirements

The area must meet the following space requirements. Allow more space to access the rear of the machine for maintenance and repairs.

- **Machine Size** 42" × 36" (1.1 m × 0.9 m)
- **Machine Height** 72" (1.8 m)



- **Typical System Footprint** 62" × 52" (1.6 m × 1.3 mm)

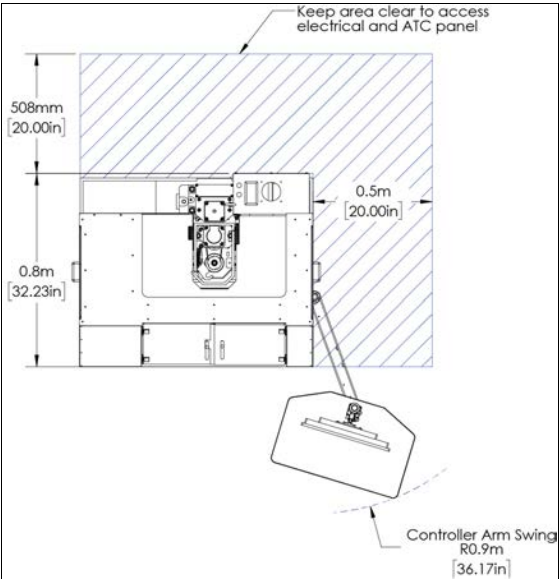


Figure 3-1: Dimensions of the machine and it's required added space, as viewed from above.

3.2 ELECTRICAL AND POWER REQUIREMENTS

You must verify that the site conforms to the following electrical and power requirements.

3.2.1 Electrical Requirements

A certified electrician must make all electrical connections, and it's your responsibility to verify that the electrical installation of the machine meets all local regulations and electrical codes.

- **Primary Power Required** Single-Phase 115 Vac, 50/60 Hz
- **Recommended Circuit Amperage** Dedicated 15 A breaker

3.2.2 Power Requirements

If the site conforms to the electrical requirements, verify that it meets the following power requirements:

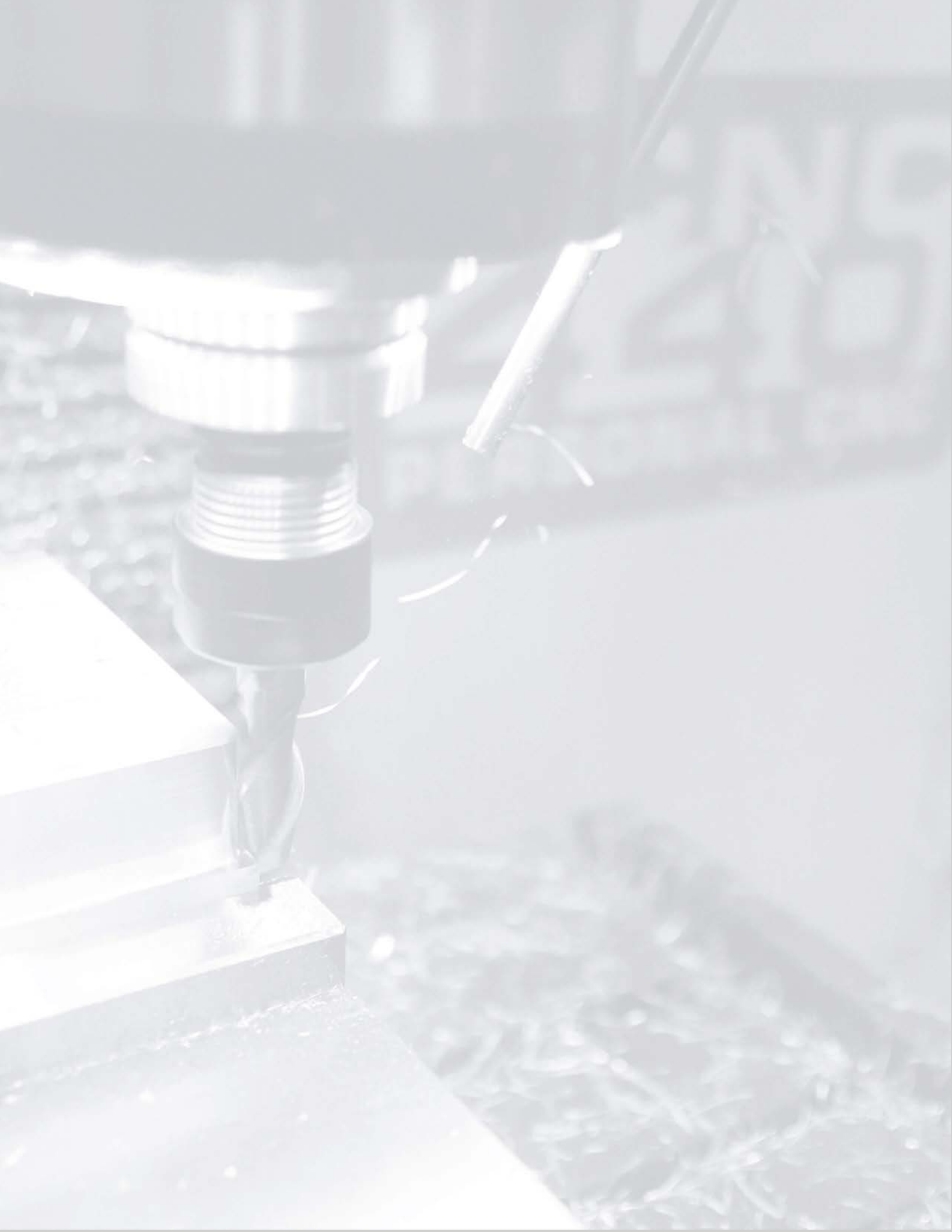
- **No Electrical Noise** Primary power must be provided by a dedicated circuit, which must be isolated from electrically-noisy devices like welders or plasma torches. The machine should be isolated from inductive loads from items like vacuum cleaners or air compressors.
- **No Ground Fault Circuit Interrupter** Power for the machine must not be protected by a ground fault circuit interrupter (GFCI), as it interferes with the operation of the BLDC spindle drive.
- **Proper Grounding** You must properly ground the power input to the machine. Examine the continuity between bare metal on the machine frame and true earth ground (a water pipe or similar) to verify that it's properly grounded.
- **Correct Plug Pattern** The machine is shipped with a NEMA 5-15P plug, designed for use with a NEMA 5-15R receptacle.

3.2.3 Options for Non-Conforming Sites

For sites that don't conform to the specified "Electrical and Power Requirements" (above), you may consider the following. You must consult with an electrician to determine the suitability for your site.

- **Step-Down Transformer** Used at sites where 115 Vac service is not possible. You must make sure that the step-down transformer has sufficient capacity to supply the machine, accessories, and any 115 Vac auxiliary

equipment. We recommend the Step-Up/Step-Down Transformer (PN 32009).



INSTALLATION

IN THIS SECTION, YOU'LL LEARN:

- About the installation process required for this machine.

The installation of the base machine takes about two hours, not including items such as a Full Enclosure Kit, a Power Drawbar, or an Automatic Tool Changer (ATC).



Note: For ease of installation, don't install the chip pans, backsplash/splash guard, enclosure, or other accessories until after you've lifted the machine onto the Machine Stand.



Before operating the machine in any way, you must read and understand this section.

CONTENTS

4.1 Before You Begin	36
4.2 Installation Tools and Items	37
4.3 Move the Pallet	38
4.4 Assemble the Machine Stand	39
4.5 Unpack the Machine Crate	43
4.6 Install Core Components	44
4.7 Install Accessory Components	52
4.8 Set Up the PathPilot Controller	82

4: INSTALLATION

4.1 Before You Begin

4.1 BEFORE YOU BEGIN

1. Inspect the item(s):

- Photograph any damage that may have occurred during shipping.
- Note any damage on the delivery receipt before signing for the shipment. If there is extensive visible damage to the machine, refuse to accept the shipment.
- Verify the received goods against the packing list.

If there is any damage or shortages, you must contact Tormach within 30 days of receipt. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.

2. If there are any items that have not yet been delivered, we recommend waiting until you've received all shipments to begin installing the machine. Depending on the product and options ordered, the system may arrive in one or more shipments of:

- Accessories (if applicable)
- Machine



Note: The machine system and large accessories are sent by freight carrier. Smaller accessories may be sent by parcel service.

4.1.1 Machine Stand

If you didn't purchase a machine stand and plan to use an alternative, you must verify that your stand:

- Has a minimum weight capacity of 1000 lb (454 kg).
- Allows for the machine to be securely connected to it.
- Is stable and secure for machining.

4.2 INSTALLATION TOOLS AND ITEMS

Before uncrating and beginning the initial setup of your machine, collect the following tools and items.

- 1000-lb capacity engine hoist



Note: Make sure that the distance between the engine hoist legs is at least 22 inches.

- Assistant to help you
- Metric hex wrench set
- Pallet jack
- Pry bar
- Safety eyewear that meets ANSI Z87+
- Shears or knife
- Socket set
- Step ladder
- Snips
- Work gloves

4: INSTALLATION

4.3 Move the Pallet

4.3 MOVE THE PALLET

Shipments arrive in crates loaded on pallets, which the freight carrier unloads onto the curb or loading dock.



WARNING! Transportation and Lift Hazard: Before moving the machine, you must confirm that all persons are clear of the area below the machine. Qualified professionals must transport, lift, and move the machine. Moving parts can entangle, pinch, or cut you, causing death or serious injury.

- Verify that the ground surface is smooth and clean of debris, and then use a pallet jack to move the pallet(s) to the desired installation location.



Note: If the ground is not smooth, you may need to use a forklift (or similar lifting equipment rated for uneven surfaces) to move the pallet(s).

4.4 ASSEMBLE THE MACHINE STAND

You can use thread-locking fluid (not provided) on the screws during the assembly procedure.

1. Before you begin assembly, we recommend that you put a large piece of cardboard (or similar) on the floor to protect the machine stand during certain steps that require access to the bottom of the stand.
2. Find the four legs (PN 51493), the left side panel (PN 51498), and the right side panel (PN 51483). Make note of the following on each component:
 - Identify the threaded holes in the legs — these are used for the door hinges. When assembled, the threaded holes must be facing forward.
 - Identify the holes for the upper shelf on the side panels. When assembled, these holes must be toward the top of the stand's legs.
3. Use three M5 × 0.8 - 10 screws (PN 38205) to attach two legs to the left side panel. Repeat this step to attach the remaining two legs to the right side panel.

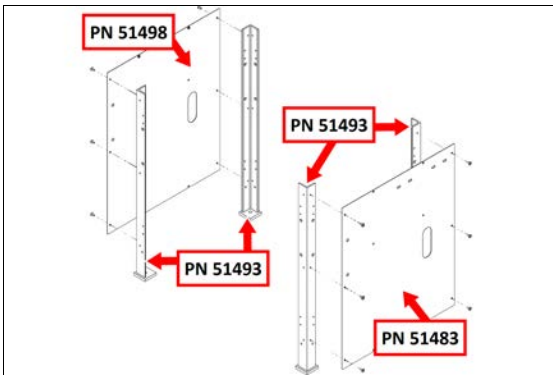


Figure 4-1: Attaching the legs to the two side panels.

4. Find the back panel (PN 51484). Use six M5 × 0.8 - 10 screws (PN 38205) to attach it to the two back legs.

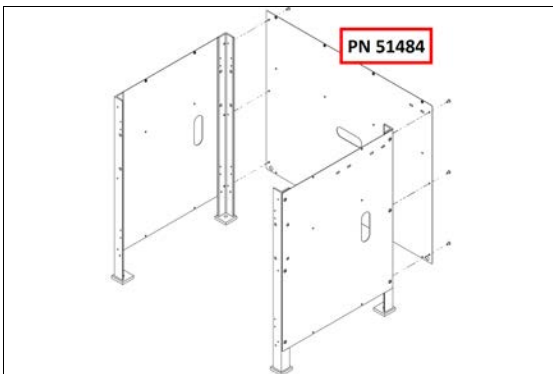


Figure 4-2: Attaching the back panel to the two side panels.

5. Find the bottom shelf (PN 51485). Use four M5 × 0.8 - 10 screws (PN 38205) to attach it to the two side panels.

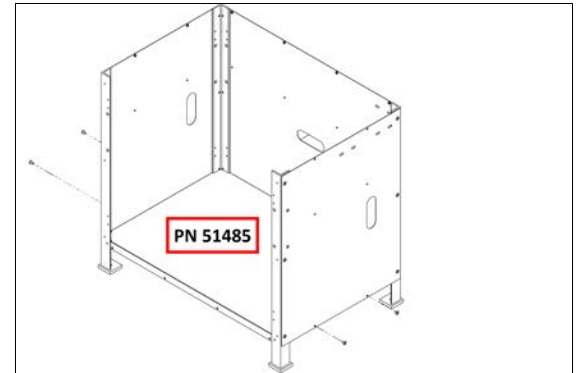


Figure 4-3: Bottom shelf attached to the two side panels.

6. Find the upper shelf (PN 51486). Use eight M5 × 0.8 - 10 screws (PN 38205) to attach it to the two side panels and the back panel.

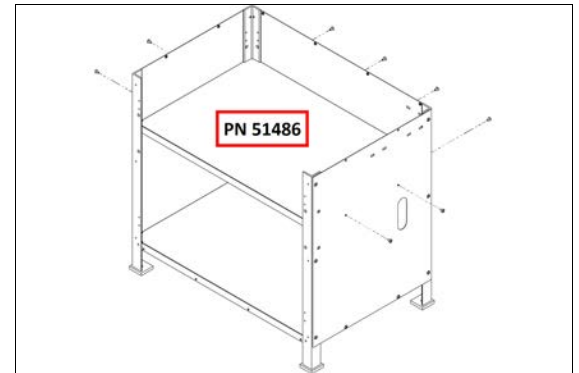


Figure 4-4: Upper shelf attached to the two side panels and the back panel.

4: INSTALLATION

4.4 Assemble the Machine Stand

7. Find the front panel (PN 51487). Use six M5 × 0.8 - 10 screws (PN 38205) to attach it to the two side panels and the bottom shelf.

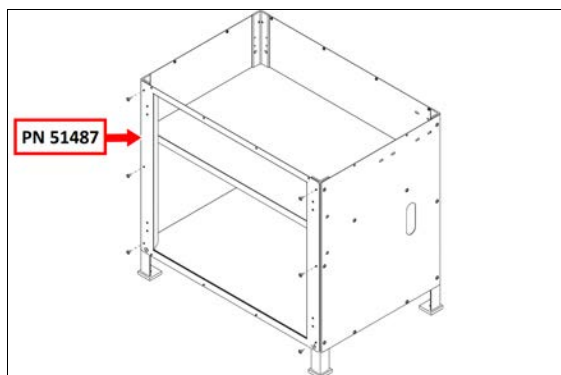


Figure 4-5: Front panel attached to the two side panels.

8. Lay the machine stand onto the cardboard on the floor.
9. Find two bottom stiffeners (PN 51494). Attach them to the bottom of the machine stand as follows:
 - a. Use six M5 × 0.8 - 10 screws (PN 38205) to attach one bottom stiffener to the two legs and the back panel of the machine stand.

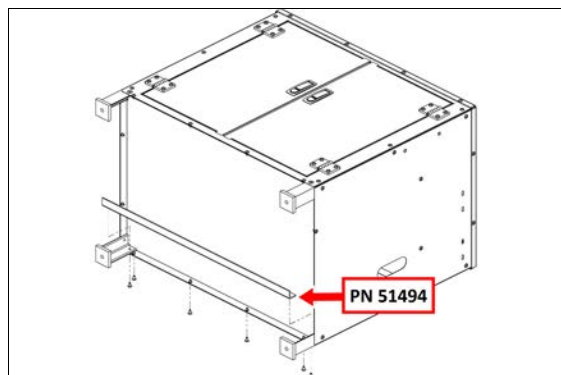


Figure 4-6: Attaching the bottom stiffener to the back of the machine stand.

- b. Use six M5 × 0.8 - 10 screws (PN 38205) to attach one bottom stiffener to the two legs at the front of the machine stand.

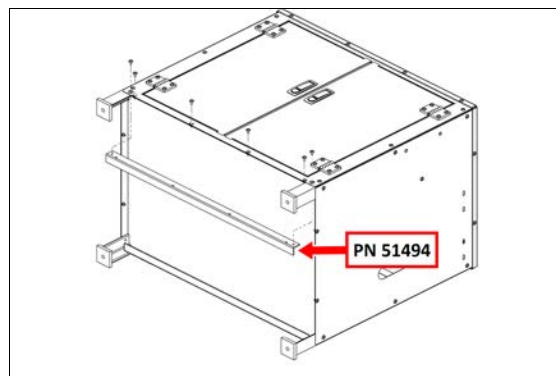


Figure 4-7: Attaching the bottom stiffener to the front of the machine stand.

10. Move the machine stand back upright.
11. Find the four door hinge assemblies (PN 51496), the left door (PN 51488), and the right door (PN 51489). Attach the doors as follows:
 - a. Use four M5 × 0.8 - 10 screws (PN 38205) to attach two door hinge assemblies to the left door.
 - b. Use four M5 × 0.8 - 10 screws to attach both door hinges to the left side of the front panel.
 - c. Repeat Steps A through B to attach the remaining two door hinges to the right door and the right side of the front panel.

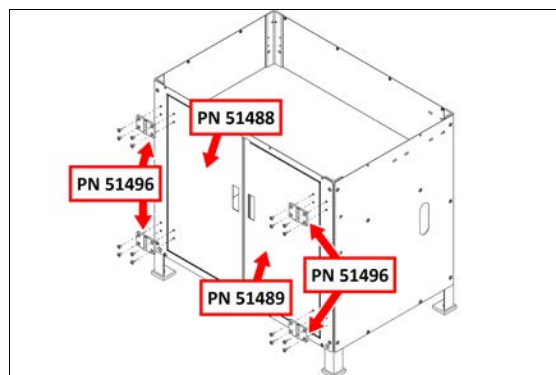


Figure 4-8: Attaching the door hinges to two doors and the front panel.

12. Find the two door latches (PN 51495). Attach them to each door as follows:
 - a. Remove the bracket from the back of the door latch, and temporarily set it aside.



Figure 4-9: Bracket on the back of the door latch.

- b. Put the door latch through the opening of the door, and reinstall the bracket that you set aside in Step A.



Figure 4-10: Attaching the door latch to the door with the bracket.

13. Find four top cover blocks (PN 51491), the top cover (PN 51490), and two top cover stiffeners (PN 51492). Assemble the top cover as follows:

- a. Align two of the top cover blocks with the pre-drilled holes on the left side of the top cover.
 - b. Align one top cover stiffener to the underside of the top cover.



Note: Make sure that the inside of the stiffener faces the middle of the top cover.

- c. Use two M12 × 20 mm socket head cap screws (PN 51503) to attach each top cover block to the top cover and the stiffener.



Note: The center hole in each top cover block is used to mount the machine to the chip pan.

- d. Repeat Steps A through C to attach the remaining top cover blocks and top cover stiffeners to the right side of the top cover.

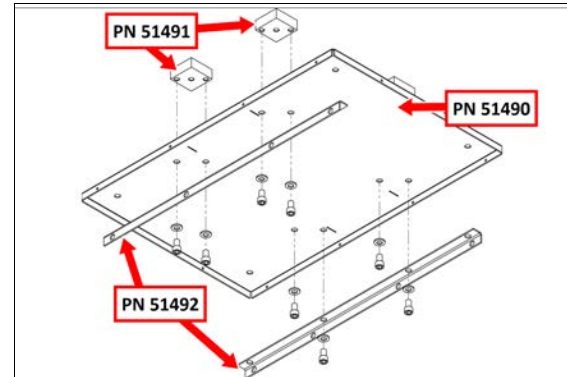


Figure 4-11: Top cover stiffener, top cover, and top cover blocks assembly.

- e. Use one M12 × 20 mm socket head cap screw (PN 51503), one M12 flat washer (PN 32473), one M12 lock washer (PN 30321), and one M12 nut (PN 35283) to secure the ends of the top cover stiffeners to each corner of the top panel.

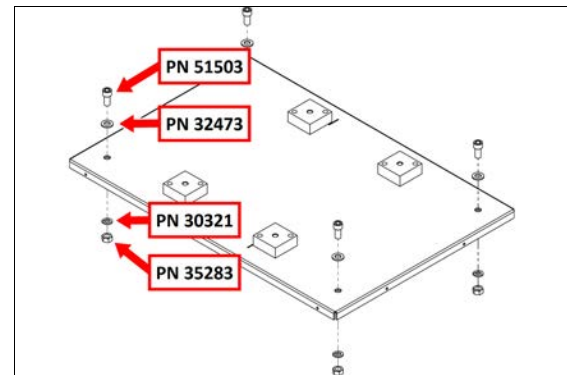


Figure 4-12: Upper support and top panel corner assembly.

4: INSTALLATION

4.4 Assemble the Machine Stand

14. Use 12 M5 × 0.8 - 10 screws (PN 38205) to attach the top cover to the stand assembly.

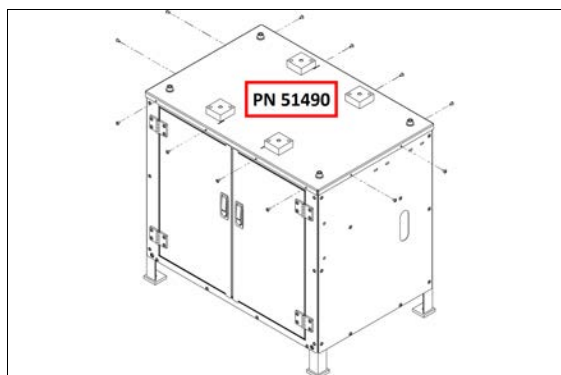


Figure 4-13: Top panel attached.

15. Lay the machine stand onto the cardboard on the floor.
16. Find four M12 nuts (PN 35283) and four leveling feet (PN 36034). Attach the feet to the machine stand as follows:
 - a. Put one M12 nut on each of the four leveling feet.
 - b. Put each leveling foot into each leg as shown in the following image. Make sure that the M12 nut is on the bottom of the leg.

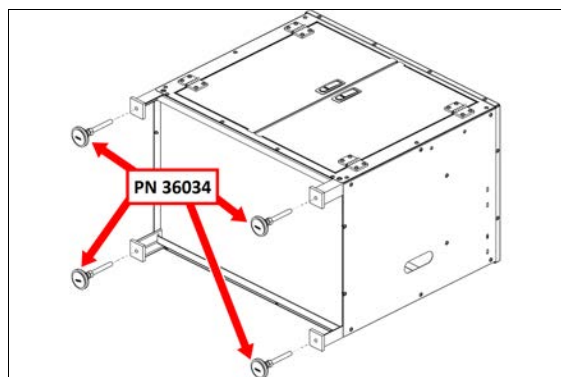


Figure 4-14: Attaching the feet.

- c. Lift the machine stand off the floor, and set it down on its feet.
- d. (Optional) Level the feet as needed.

4.5 UNPACK THE MACHINE CRATE



CAUTION! Sharp Objects Hazard: Before opening the shipping crate, you must put on work gloves and safety eyewear that meets ANSI Z87+. If you don't, the shipping crate and steel straps could cut you, causing serious injury.

1. Put on work gloves and eye protection.
2. Cut and remove the steel straps on the shipping crate with snips.
3. Disassemble the shipping crate with a hammer and pry bar. Start with removing the top, followed by the four sides.
4. Remove and discard the plastic wrap from the machine.
5. Inspect the item(s):
 - Photograph any damage that may have occurred during shipping.
 - Verify the received goods against the packing list.If there is any damage or shortages, you must contact Tormach within 30 days of receipt. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.
6. Cut and remove the shipping straps securing the tool box to the pallet.

The tool box contains the following items:

 - Drawbar
 - Drawbar thrust washer
 - Electrical cabinet key
 - Hex wrench set: 3 mm, 4 mm, 5 mm, 6 mm, 8 mm, and 10 mm
 - Phillips screwdriver
 - Tormach Tooling System (TTS) collet
7. Cut and remove the shipping straps securing the X-axis motor cover plate and X-axis limit switch cover plate from the pallet.

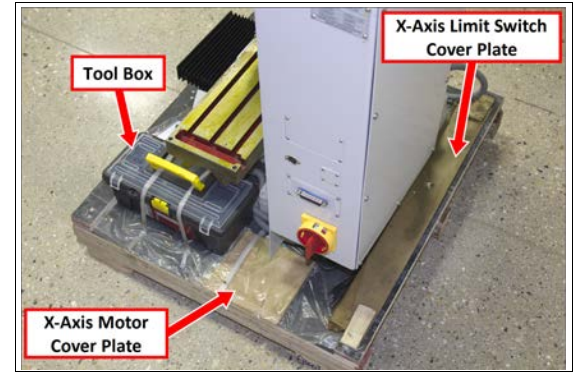


Figure 4-15: Components on the pallet.

8. Cut the shipping straps securing the operator box to the pallet.
9. Remove all packing materials from the spindle head.

4: INSTALLATION

4.6 Install Core Components

4.6 INSTALL CORE COMPONENTS

After moving and unpacking the machine crate, you must install the core components of the system. Complete the following steps in the order listed:

4.6.1 Lift and Move the Machine.....	44
4.6.2 Install the Emergency Stop Box.....	45
4.6.3 Install Cover Plates.....	45
4.6.4 Install the Controller.....	45
4.6.5 Verify the Installation.....	48

4.6.1 Lift and Move the Machine

- 1. Move the machine's pallet and stand as close as possible to the desired location with a pallet jack. The machine stand must be positioned behind the machine.
- 2. Put the chip pan on top of the machine stand.
- 3. Remove any excess shipping grease on the machine with a mild degreaser (like WD40® or similar).
- 4. Remove the nuts from the four studs that hold the machine to the pallet with a 16 mm wrench. Discard the nuts.

**WARNING!** Transportation and Lift Hazard:
Before moving the machine, you must confirm that all persons are clear of the area below the machine. Qualified professionals must transport, lift, and move the machine. Moving parts can entangle, pinch, or cut you, causing death or serious injury.

- 5. Secure an engine hoist to the eye bolt on the top of the machine column, and then lift the machine from above.



Figure 4-16: Engine hoist secured to the ey

**WARNING!** Crush Hazard: While lowering the machine onto the machine stand, you must keep your hands away from all moving parts. Moving parts can entangle, pinch, or cut you, causing death or serious injury.

- 6. Slowly and steadily lower the machine on to the chip pan. Confirm that the holes in the chip pan, machine, and machine stand are aligned.



Figure 4-17: Lowering the machine onto the stand.

7. Secure the machine to the machine stand with a 10 mm hex wrench and four M12 × 100 mm socket head cap screws, four M12 washers, and four M12 nuts.
8. Once the machine is completely supported by the machine stand, remove the engine hoist.

4.6.2 Install the Emergency Stop Box

1. Identify the emergency stop mounting bracket (PN 53003) and two M6 flanged button head cap screws (PN 38206) included with the chip pan.
2. Install the emergency stop mounting bracket onto the right side of the chip pan with a 4 mm hex wrench.

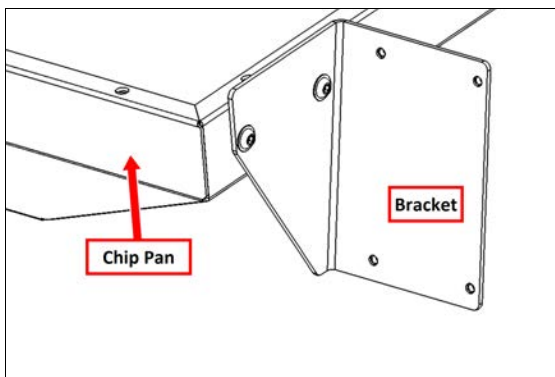


Figure 4-18: Emergency stop mounting bracket installed on the chip pan.

3. Identify the four M4 button head cap screws (PN 53009) included with the chip tray and emergency stop box connected to the machine's electrical cabinet.
4. Remove and set aside the four screws securing the lid of the emergency stop box with a Phillips screwdriver. Remove and set aside the lid.



Note: There are wires attached to the lid. Be careful not to pull on them.

5. Insert one M4 screw into each corner of the emergency stop box so that the threads of the screw stick out from the back of the box.
6. Attach the emergency stop box to the emergency stop mounting bracket with a 2.5 mm hex wrench.
7. Re-install the emergency stop box lid with the screws that you set aside earlier.

4.6.3 Install Cover Plates

1. Find the X-axis motor cover plate and the X-axis limit switch cover plate that you set aside in "Unpack the Machine Crate" (page 43).
2. Attach the X-axis motor cover plate to the X-axis motor mount with two screws.
3. Attach the X-axis limit switch cover plate to the front of the machine table with two screws.

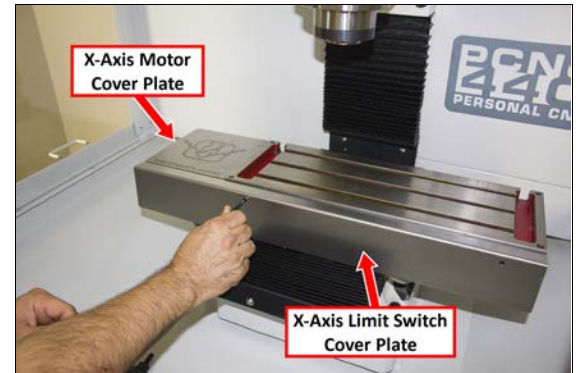


Figure 4-19: Cover plates installed on the machine table.

4.6.4 Install the Controller

Install the Controller Arm

Tools and Items Required

- 4 mm hex wrench
- 6 mm hex wrench
- 8 mm hex wrench
- 17 mm hex wrench
- 17 mm socket wrench
- 21 mm wrench
- Dead-blow hammer (or similar)
- Phillips screwdriver
- Pry bar



CAUTION! Sharp Objects Hazard: Before opening the shipping crate, you must put on work gloves and safety eyewear that meets ANSI Z87+. If you don't, the shipping crate and steel straps could cut you, causing serious injury.

1. Put on work gloves and eye protection.
2. Open the Controller Arm crate with a pry bar.

4: INSTALLATION

4.6 Install Core Components

- Secure the square tube arm to the machine stand with two M8 socket head cap screws, two M8 flat washers, two M8 lock washers, and a 6 mm hex wrench. Verify that the white nylon washer is located toward the bottom of the mounting pad.

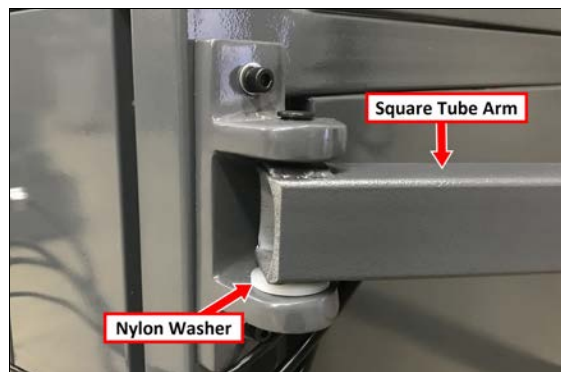


Figure 4-20: White nylon washer on the square tube arm.

- Put the monitor post into the square tube arm. Verify that the monitor bracket is toward the top, and that the threaded holes face the holes in the square tube arm.
- Tighten the cross bolt on the square tube arm with a 17 mm socket wrench and a 17 mm hex wrench.
- With a 21 mm wrench, remove the monitor bracket from the Controller Arm, and rotate it so that the largest mounting plate is facing up.



Figure 4-21: Incorrect and correct orientations of the monitor bracket on the Controller Arm.



Note: The largest mounting plate is for the monitor, and the smallest mounting plate is for the keyboard tray.

- Tighten the three pivot bolts on the monitor bracket with an 8 mm hex wrench and a 16 mm wrench.



Figure 4-22: Tightening the monitor bracket in place.



Tip! This makes it easier to install the monitor, which you'll do later in this installation procedure.

- Tap the end plug into the square tube arm with a dead-blow hammer (or similar).
- Secure the keyboard table to the monitor bracket with four M5 socket head cap screws, four M5 flat washers, four M5 split lock washers, and a 4 mm hex wrench.
- Attach four wire tie mounts to the monitor post with four 4 mm flat head machine screws and a Phillips screwdriver.

Install the Monitor

The PathPilot controller mount allows you to install the PathPilot controller behind the monitor (which is attached to the Controller Arm).



Note: If you're using a Touch Screen Kit (PN 35575), you must first remove the stock mounting bracket from the back of the monitor.

1. Put the PathPilot Controller VESA Mount (PN 50382) against the monitor mounting plate. Then, put the monitor on the other side of the PathPilot controller mount, and align the holes on the three components.
2. Attach the monitor, PathPilot Controller VESA Mount, and monitor mounting plate together with four M4 × 12 mm socket head cap screws (provided with the PathPilot Controller VESA Mount).

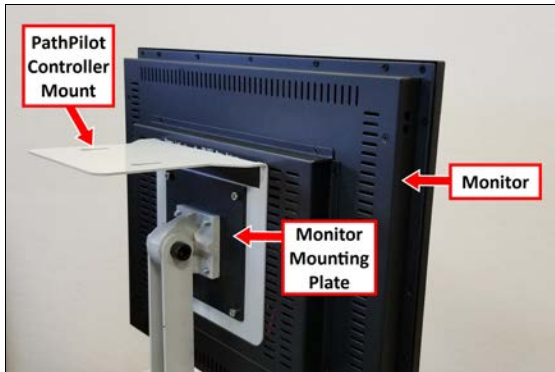


Figure 4-23: PathPilot controller mount attached to the back of the monitor.

3. Adjust the position of the monitor and the keyboard tray with an 8 mm hex wrench and a 16 mm wrench. Once complete, securely tighten the pivot screws.

Install the PathPilot Controller

The PathPilot controller attaches to the top of the PathPilot controller mount and behind the monitor.

1. Put four standoffs into the controller and tighten them by hand.



Figure 4-24: Installing the standoffs on the controller.

2. Find the four M4 screws and the VESA plate included with the controller. Then, mount the VESA plate to the PathPilot Controller VESA Mount (PN 50382). Make sure to put it flat side down with the keyholes toward the monitor.



Figure 4-25: VESA plate mounted onto the PathPilot Controller VESA Mount (PN 50382).

3. Attach the controller to its mount by sliding the standoffs through the key slots.



Figure 4-26: Controller attached to the top of the mount.

4. Find the Machine Interface Adapter provided. Then, remove the film from the double-sided tape on its case.



Figure 4-27: Adapter identification.

4: INSTALLATION

4.6 Install Core Components

5. Put the case below the PathPilot Controller VESA Mount. Verify that the case is oriented so that the DB-25 port is facing to the left side of the monitor from behind. Press up firmly to attach it in place.

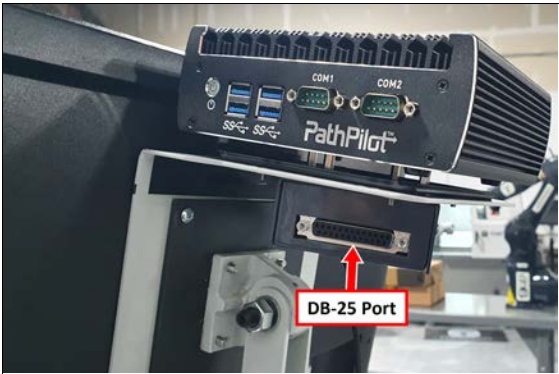


Figure 4-28: Adapter attached to the bottom of the mount.

Note: You may have to mount the case at an angle to fit between any screws on the PathPilot Controller VESA Mount.

6. Connect all USB accessories to the controller:
- Jog Shuttle (PN 30616) (Optional)
 - Keyboard
 - Mouse
 - Monitor
7. Connect the monitor's video cable to the controller.
8. Route the loose end of the video cable toward the monitor, and then connect it to the monitor.
9. Connect the power supply to the controller.
10. Route the loose end of the power cords from both PathPilot controller units to a power strip, and then connect them.
11. Find the DB-25 interface cable provided with this kit. Connect the DB-25 interface cable into the DB-25 outlet on the side of the electrical cabinet.
12. Route the loose end of the DB-25 interface cable to the Machine Interface Adapter, and then connect it to the DB-25 port.
13. Find the Ethernet cable, and then connect it to the Ethernet outlet on the Machine Interface Adapter.
14. Route the loose end of the Ethernet cable toward the controller and then connect it.

15. Secure the wire loom, Ethernet cable, and power supply cables to the Controller Arm with six wire tie mounts and six cable ties.

4.6.5 Verify the Installation

To properly validate the core installation of your machine, you must understand how to power on and off the machine and use the controls.

Complete the following steps in the order listed:

Power on the Machine	48
Verify Limit Switch Function	49
Verify Axes Function	50
Verify Spindle Function	50
Power off the Machine	51

Power on the Machine

1. Use a multimeter to verify that the electrical service in your location meets the following requirements. If your location does not meet these requirements, do not install the machine. Instead, you must consult with a local electrician about your options.
- **Primary Power Required** Single-Phase 115 Vac, 50/60 Hz
 - **Recommended Circuit Amperage** Dedicated 15 A breaker
2. Connect the machine's mains power cable to the verified electrical service.
3. Find the Main Disconnect switch, and then remove the hang tag.

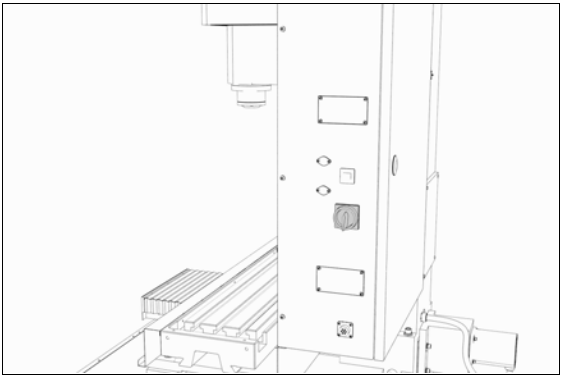


Figure 4-29: The Main Disconnect switch on the side of the machine's electrical cabinet.

4. Push the Power button on the power strip.

- Turn the Main Disconnect switch to **ON**.

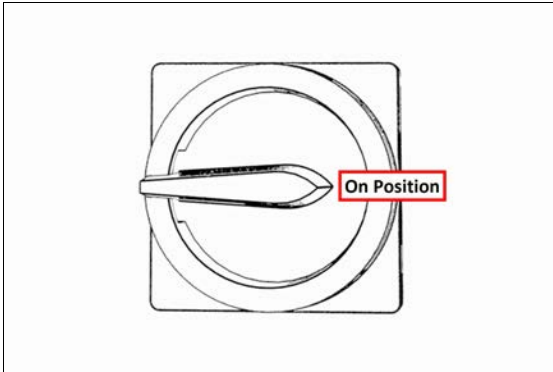


Figure 4-30: Example of the Main Disconnect switch in the On position.

Mains power is now connected to the machine.

- Push the Power button on the PathPilot controller, if it's not already powered on.
- Rotate the Emergency Stop button one-quarter turn clockwise to release it.



Figure 4-31: Emergency Stop button.

- Push the green Start button to enable the machine.



Figure 4-32: Reset button.

The axis drives are now powered on.

- From the PathPilot interface, on the **Status** tab, verify that the **Machine OK** light changes from yellow to green.

The machine is powered on and ready to operate.

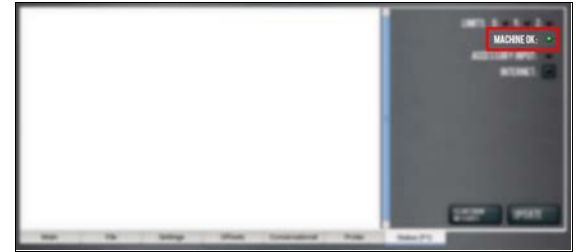


Figure 4-33: Machine OK light on the Status tab.

- Select **Reset**.



Figure 4-34: Reset button.

This initializes the connection between the machine and controller.

Verify Limit Switch Function

You must confirm that the limit switches correctly operate.

- Power on the machine and the PathPilot controller.
 - Turn the Main Disconnect switch to **ON** on the side of the electrical cabinet.
 - Twist out the machine's red Emergency Stop button, which enables movement to the machine axes and the spindle.
 - Press the button.
- From the PathPilot interface, on the **Status** tab, identify the **X Limit**, **Y Limit**, and **Z Limit** lights.



Figure 4-35: X Limit, Y Limit, and Z Limit lights on the Status tab.

4: INSTALLATION

4.6 Install Core Components

- On the machine, identify the **X**, **Y**, and **Z** limit switches.

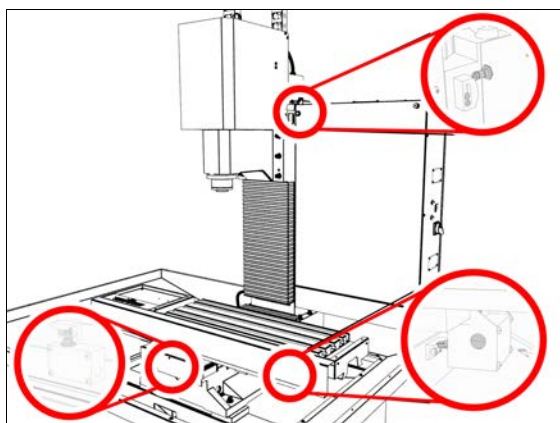


Figure 4-36: Machine limit switches.

- One at a time, press the **X**, **Y**, and **Z** limit switches. From the PathPilot interface, on the **Status** tab, the corresponding **X Limit**, **Y Limit**, and **Z Limit** lights come on.
- From the PathPilot interface, select **Reset**.

Verify Axes Function

You must confirm that the axes correctly operate.

- Power on the machine and bring it out of Reset.
- Select the **Page Up** key on the keyboard to move the spindle head up (Z+).
- Remove and discard the shipping block from the machine table.
- Reference the axes: from the PathPilot interface, select **Ref Z**, **Ref X** and **Ref Y**.



Figure 4-37: Ref Z, Ref X, and Ref Y buttons.

The machine moves to the reference position.

- Use the keyboard to verify axes motion:
 - Select the **Right Arrow** key and then the **Left Arrow** key. The table moves left (X+), then right (X-).
 - Select the **Down Arrow** key and then the **Up Arrow** key. The table moves away from you (Y-), then toward you (Y+).

- Select the **Page Down** key and then the **Page Up** key. The spindle head moves down (Z-), then up (Z+).
- If applicable, verify the optional Jog Shuttle:
 - Press any axis button on the Jog Shuttle — **X**, **Y**, **Z**, or **A** — to select an axis.



Figure 4-38: Functions on the optional Jog Shuttle.

From the PathPilot interface, on the **Main** tab, the corresponding green **Axis** light comes on.

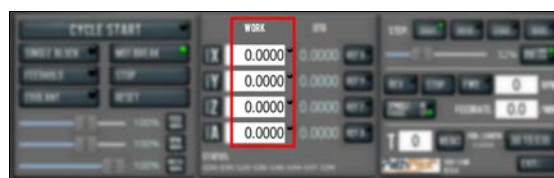


Figure 4-39: Axis lights.

- Turn the **Jog Shuttle Ring** in any direction to move the selected axis, then turn it in the opposite direction to reverse the direction.

Verify Spindle Function

You must confirm that the spindle correctly operates.

- Open the spindle door and locate the spindle belt.
- From the PathPilot interface, confirm that the **Spindle Range** button matches the spindle belt position.



Figure 4-40: Spindle Range button.

If necessary, select **Spindle Range** to match the spindle's belt position.

- Close and latch the spindle door.

- From the PathPilot interface, confirm that the **Spindle Override** slider isn't set to 0% (when it's set to 0%, the slider is yellow). To clear the override, select **RPM 100%**.



Figure 4-41: Spindle Override slider.

Note: The **Spindle Override** slider changes the programmed spindle speed by a specific percentage. If it's set to 0%, the spindle won't move during the following steps in this procedure. For information, see "About Spindle Override" (page 135).

- From the PathPilot interface, in the **RPM DRO** field, type 500. Then select the **Enter** key.
- Select **FWD**.

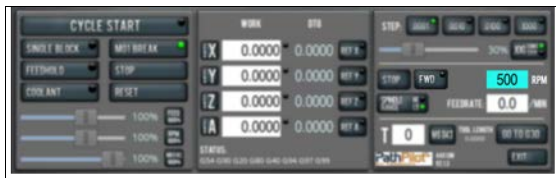


Figure 4-42: Spindle controls.

The spindle rotates clockwise at 500 rpm.

- Select **STOP**.
The spindle stops rotating.
- Select **STOP**.
The spindle stops rotating.

Power off the Machine

- Push the Emergency Stop button to lock it into the disabled position.



Figure 4-43: Emergency Stop button.

With the Emergency Stop button in the disabled position all motion and spindle function stops, the Start button is disabled. From the PathPilot interface, on the **Status** tab, the **Machine OK** light illuminates yellow.

- From the PathPilot interface, select **Exit**.
- When prompted, select **OK**.
- Once the PathPilot interface indicates that it's safe to power off the machine, turn the Main Disconnect switch to **OFF**.

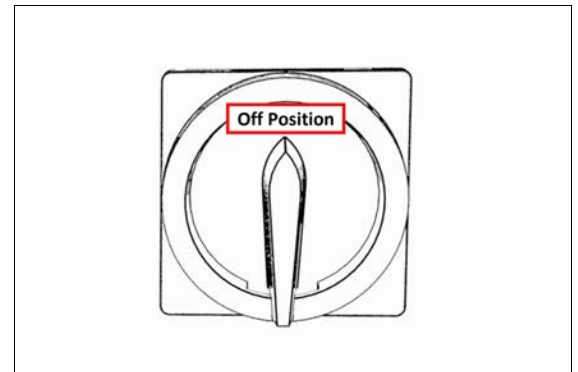


Figure 4-44: Example of the Main Disconnect switch in the Off position.

Mains power is disconnected from the machine.

4: INSTALLATION

4.7 Install Accessory Components

4.7 INSTALL ACCESSORY COMPONENTS

Install accessory components in the order specified, skipping any components that you don't have.

4.7.1 Install the Power Drawbar.....	52
4.7.2 Install the Flood Coolant Kit	59
4.7.3 Install the PCNC 440 Automatic Tool Changer.....	61
4.7.4 Install the Enclosure.....	71
4.7.5 Door Switch Installation	77

4.7.1 Install the Power Drawbar

Before You Begin

Required Tools

This procedure requires the following tools. Collect them before you begin.

- Adjustable wrench (two)
- Clean cloth
- Flat-blade screwdriver
- Metric hex wrench set
- Paint pen
- Phillips screwdriver

Air Requirements

- **Air Pressure** 90-120 psi (620-825 kPa)
If the air supply is more than 120 psi (825 kPa), you must use a regulator.
- **Air Volume** For successive, back-to-back tool changes, at least 2 cfm at 90 psi is required.
Exact air volume depends on the frequency of tool changes.
- **Dry Air** We recommend using a compressed air dryer, desiccator, or filter between the air compressor and the machine.

Installation

Prepare the Machine

1. If there's already a Tormach Tooling System (TTS) tool holder in the spindle, remove it.
2. Verify that you can access the inside of the spindle cabinet: jog the Z-axis down (-Z) until it is about 6 in. (15 cm) from the machine table.
3. Power off the machine and the PathPilot controller.

- a. Push in the machine's red Emergency Stop button, which removes power to motion control.
- b. From the PathPilot interface, select **Exit**.
- c. Turn the Main Disconnect switch to **OFF** on the side of the electrical cabinet.

Disassemble the Original Power Drawbar



CAUTION! Loose Objects Hazard: The original drawbar contains spring-loaded hardware, which could quickly become loose, causing serious injury. Before disassembling the original drawbar, you must put on safety eyewear that meets ANSI Z87+.

1. Put on safety eyewear.
2. Open the spindle door.
3. While using one hand to support the Tormach Tooling System (TTS) collet, use the other to turn and remove the original drawbar.
4. Set the collet and the original drawbar aside.
5. While using one hand to hold the spindle lock arm, use the other to remove the spindle lock arm pivot screw with a 6 mm hex wrench.

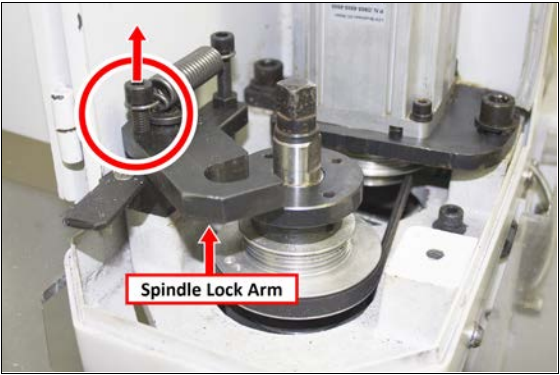


Figure 4-45: Spindle lock arm pivot screw removed from the original drawbar assembly.



Note: The spindle lock arm assembly contains spring-loaded hardware, which could quickly become loose.

6. Remove the spindle lock arm pivot bolt with a 6 mm hex wrench. Then, remove the spindle lock arm.

7. Remove the spindle lock base and its components with a 5 mm hex wrench.



Figure 4-46: Spindle lock base from the original drawbar assembly.



Note: Keep all components for future use. The original drawbar, drawbar bushing, and spindle lock are all required to operate the machine if you later decide to remove the Power Drawbar.

8. Wipe all exposed drawbar mounting surfaces inside the spindle motor cabinet with a clean cloth.



Figure 4-47: Mounting surfaces to clean inside the spindle motor cabinet.

Assemble the Power Drawbar

1. Find the drawbar that you set aside in Disassemble the Original Drawbar.
2. Remove the drawbar bushing from the drawbar, and set it aside.
3. Put Anti-Seize (provided) on the bottom of the drawbar head.
4. Find the eight spring washers provided.
5. Move one spring washer on the drawbar, with the convex side of the spring washer toward the drawbar

head.

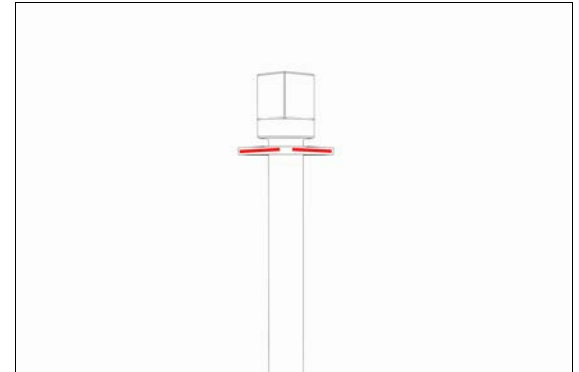


Figure 4-48: Convex side of the spring washer toward the drawbar head.

6. Move another spring washer on the drawbar, with the concave side of the spring washer toward the concave side of the spring washer from Step 5.

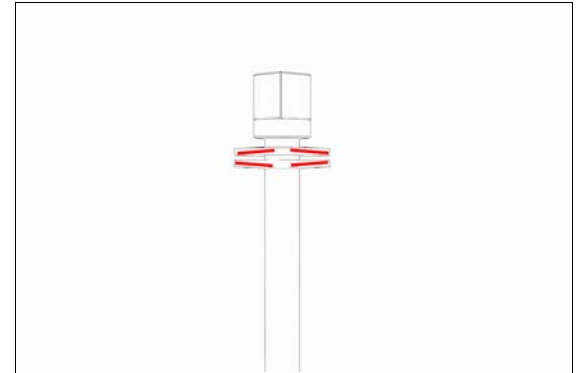


Figure 4-49: Concave sides of two spring washers together.

7. Put Anti-Seize on the edge of the contact surface between the pair of spring washers that you put on the drawbar in Steps 5 and 6.
8. Repeat Steps 5 through 7 for the remaining six spring washers. Make sure that you put Anti-Seize on the spring washers at every contact point.

4: INSTALLATION

4.7 Install Accessory Components

- Examine the stack of spring washers. Make sure that all eight spring washers are on the drawbar and arranged in four sets of opposing pairs.

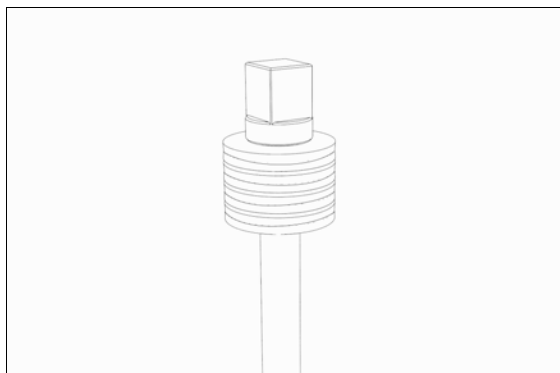


Figure 4-50: All eight spring washers on the drawbar.

- Find the drawbar bushing that you set aside in Step 2, and put it on the drawbar. Make sure that the smaller diameter of the drawbar bushing is toward the bottom of the drawbar.
- Put Anti-Seize on the top of the drawbar bushing.
- Put Anti-Seize on the bottom threads of the drawbar.

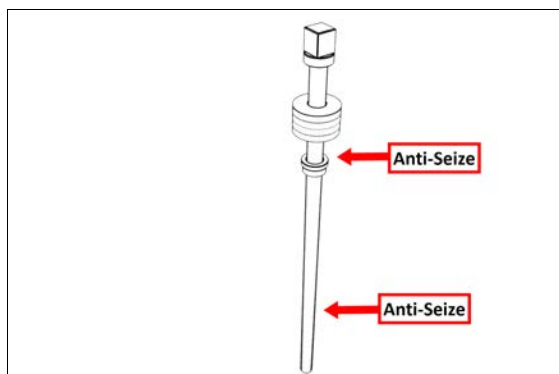


Figure 4-51: Locations to apply Anti-Seize on the drawbar assembly.

- Put Anti-Seize on the outside taper of the Tormach Tooling System (TTS) collet. Make sure that there is no Anti-Seize on the inside of the collet.
- Put the drawbar assembly into the spindle.
- While using one hand to insert the collet into the spindle, use the other to turn the drawbar into the collet.

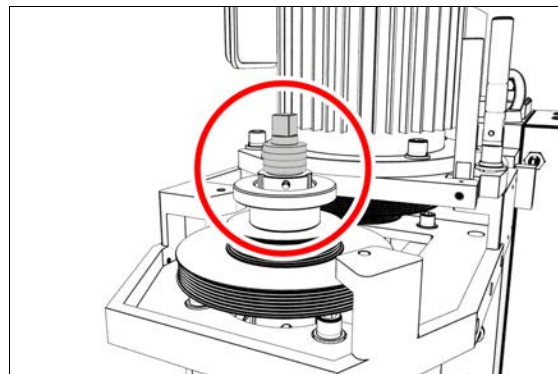


Figure 4-52: Drawbar assembly installed in the spindle motor cabinet.

- Hand-tighten the drawbar.
- While using one hand to insert an empty Tormach Tooling System (TTS) tool holder into the collet, use the other to tighten the collet into the drawbar with an adjustable wrench.
- Use two large, adjustable wrenches to tighten the Power Drawbar until the spring washer stack is compressed to a height of 3/4 in. (20 mm).

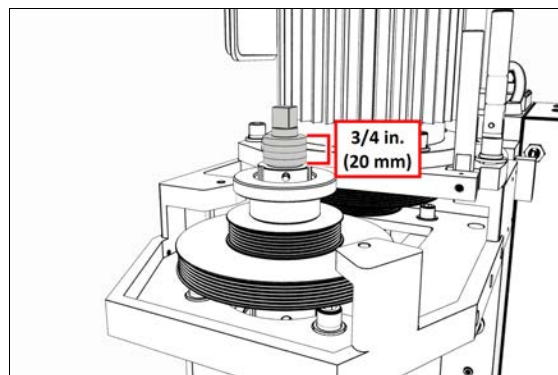


Figure 4-53: Correctly compressed spring washer stack.

Note: Later in this procedure, after you complete the initial installation, you'll do a final drawbar tension adjustment. For more information, see Adjust the Drawbar Tension.

- Find the Power Drawbar cylinder assembly provided.

20. Identify the Power Drawbar mounting surfaces inside of the spindle motor cabinet, as shown in the following image.

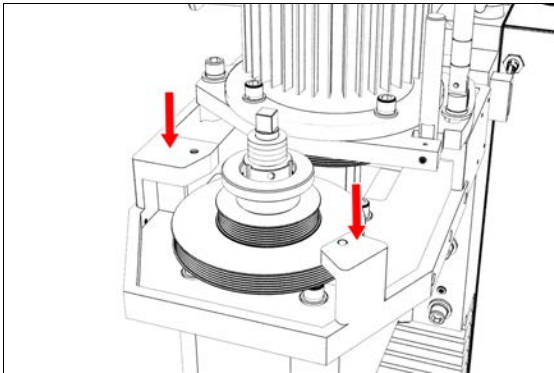


Figure 4-54: Power Drawbar mounting surfaces inside of the spindle motor cabinet.

21. Put the Power Drawbar cylinder assembly on the mounting surface.
22. Find the M10 × 16 mm shoulder screw provided, and then put Anti-Seize on its threads.
23. Install the M10 × 16 mm shoulder screw on the Power Drawbar cylinder's mount plate with a 5 mm hex wrench.

Note: The head of the shoulder screw will protrude above the mounting plate when the thread of the shoulder screw is fully installed.

The unit, which floats by design, is now securely installed to the spindle head.



Figure 4-55: M10 × 16 mm shoulder screw installed on the Power Drawbar cylinder's mount plate.

24. Find the quick-release pin provided, and then insert it into the Power Drawbar cylinder's mount plate. The quick-release pin allows the Power Drawbar cylinder to pivot within the spindle motor cabinet.

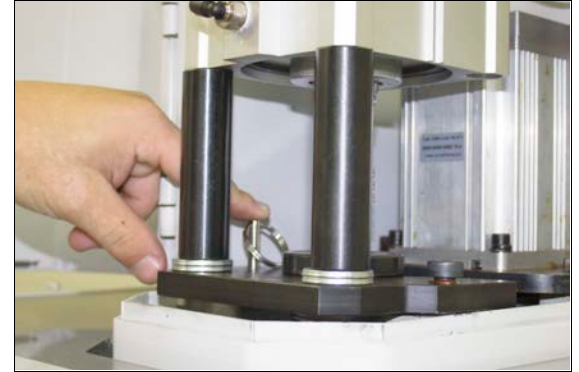


Figure 4-56: Quick-release pin inserted into the Power Drawbar cylinder's mount plate.

Make Air Connections



Note: If you have an (optional) Automatic Tool Changer (ATC), install it now.

1. Remove two M5 × 15 mm socket head cap screws securing the coolant hose mount bracket to the mill head with a 4 mm hex wrench. Then, set aside the coolant hose mount bracket.



Figure 4-57: Coolant hose mount bracket on the mill head.

4: INSTALLATION

4.7 Install Accessory Components

- Find the button box assembly provided, which includes the following:

Quantity	Description
6 ft	Tube, 1/4 in. OD, Nylon (PN 31457)
8 ft	Tube, 5/32 in. OD, Nylon (PN 35922) Note: You'll need two pieces of this tube to make the air connections. Cut it in to two 4 ft pieces before connecting the power drawbar button to the cylinder.
1	Bracket (PN 37333)
1	Button Box Base (PN 37291)
1	Button Box Cover (PN 37292)
2	Nut, Hex, M5 × 0.8 (PN 31201)
7	M5 × 6 mm Phillips Head Machine Screw (PN 37299)
2	Screw, Socket Head Cap, M5 × 0.8 - 12 mm (PN 31353)
1	Power Drawbar Button (PN 37296)

- Install the button box base on the mill head with two M5 × 6 mm Phillips head machine screws. Make sure that it's flush with the bottom of the spindle cabinet, as shown in the following image.



Figure 4-58: Button box base installed on the mill head.

- Remove the push button cap and the O-ring from the Power Drawbar button.



Figure 4-59: Power Drawbar button.

- Put the Power Drawbar button in the button box base, and then secure it in place with the O-ring and the push button cap that you removed in Step 4.
- Find the coolant hose mount bracket that you set aside in Step 1. Then, install it to the button box cover with two M5 × 12 mm socket head cap screws and two M5 nuts. Set aside the button box cover assembly.

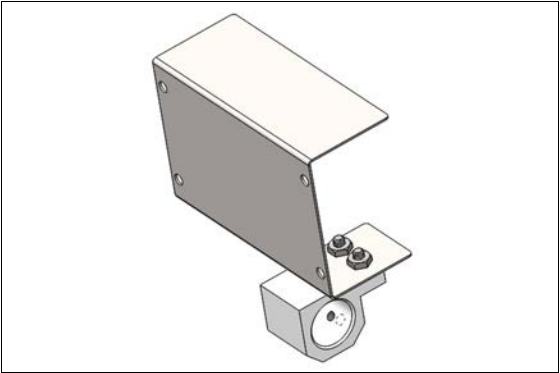


Figure 4-60: Coolant hose mount bracket installed on the button box cover.

- Connect one end of the 5/32-in. plastic tube to the Retract port on the Power Drawbar button.

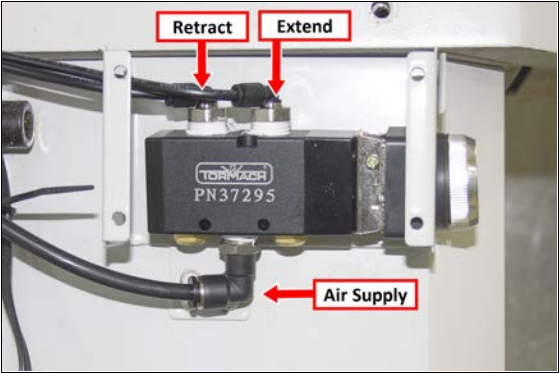


Figure 4-61: Power Drawbar button air line routing.

8. Route the loose end of the Retract air line to the Power Drawbar cylinder, and then connect it to the lowest elbow fitting.

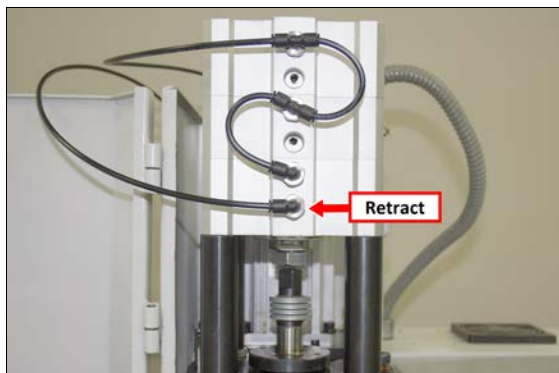


Figure 4-62: Retract fitting on the Power Drawbar cylinder.

9. Connect one end of the remaining 5/32-in. plastic tube to the Extend port on the Power Drawbar button.
10. Route the loose end of the Extend air line to the Power Drawbar cylinder, and then connect it to the top tee fitting.

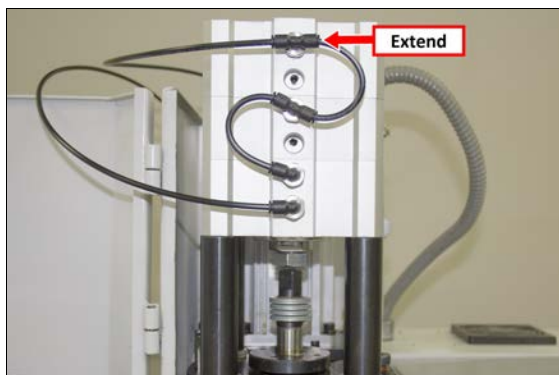


Figure 4-63: Extend fitting on the Power Drawbar cylinder.

11. Use the five cable ties to secure both air lines together.
12. Connect one end of the 1/4-in. plastic tube to the Air Supply push-to-connect elbow on the Power Drawbar button.
13. Route the loose end of the 1/4-in. plastic tube out of the enclosure and toward the air compressor.
14. Put the 1/4 in. push-connect air fitting adapter on the loose end of the 1/4 in. plastic tube.
15. Route the 1/4 in. plastic tube to your shop's air supply and connect it with the adapter.

16. Replace the button box cover that you set aside in Step 6 with a Phillips screwdriver and four M5 × 6 mm Phillips head machine screws.
17. Remove the M6 × 12 mm socket head cap screw from the mill head and set it aside.



Figure 4-64: M6 × 12 mm socket head cap screw to remove.

18. Install the bracket on the button box cover with one M5 × 6 mm Phillips head machine screw.
19. Install the opposite side of the bracket to the mill head with the M6 × 12 mm socket head cap screw that you set aside in Step 17.

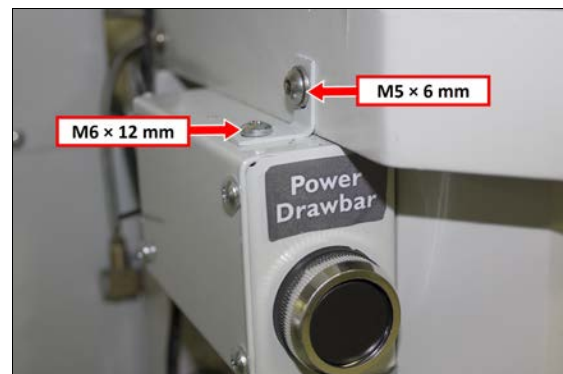


Figure 4-65: Bracket installed on the Power Drawbar button box and the mill head.

Adjust the Power Drawbar

NOTICE! After the initial installation, you must complete a final drawbar tension adjustment. For more information, see Adjust the Drawbar Tension. If you don't complete a drawbar tension adjustment, there's a risk of tool pull-out.

4: INSTALLATION

4.7 Install Accessory Components

Adjust the Drawbar Tension

This adjustment sets the highest possible drawbar tension while still allowing the Power Drawbar cylinder to release the tool. For information, see About Drawbar Tension.

NOTICE! After the initial installation, you must examine the drawbar tension weekly. During periods of heavy use, examine the drawbar tension more frequently. If you don't, there's a risk of tool pull-out.

To adjust the drawbar tension:

1. Put an empty Tormach Tooling System (TTS) tool holder into the collet.
2. While using one hand to support the tool holder, use the other to push the Release Tool button.
3. Depending on whether the tool holder releases or not, do one of the following:
 - **If the Tool Holder Releases** Tighten the Power Drawbar in quarter-turn increments with two adjustable wrenches. After each turn, push the Release Tool button. Stop when the tool holder does not release. Then, loosen the Power Drawbar one quarter-turn with two adjustable wrenches.
 - **If the Tool Holder Doesn't Release** Loosen the Power Drawbar in quarter-turn increments with two adjustable wrenches while pushing the Release Tool button. Stop when the tool holder releases.
4. Make a visual reference to help you set or adjust the drawbar tension in the future: use a paint pen to make a witness mark on both the head of the drawbar and the end of the spindle.

About Drawbar Tension

While machining, the Tormach Tooling System (TTS) collet holds a Tormach Tooling System (TTS) tool holder in the spindle by applying a clamping force to both the shank and the shoulder of the tool. The tension force that is applied to the drawbar pulls the Tormach Tooling System (TTS) collet into the spindle taper, which then applies the clamping force to the Tormach Tooling System (TTS) tool.

The force on the drawbar — known as drawbar tension — is applied differently depending on the tool changing method:

- **Automatic (using the Power Drawbar)** Tension is applied by the compressed spring washers.

- **Manual** Tension is applied when you tighten the drawbar into the collet using a wrench.

Adjust the Initial Setup

In this adjustment, you'll verify that there's enough clearance between the end of the drawbar and the Power Drawbar cylinder.

NOTICE! If you don't do this adjustment, there's a risk that the drawbar can loosen, or that operations can be louder than normal.

1. Examine the space between the hex head screw on the Power Drawbar cylinder's rod and the top of the drawbar.

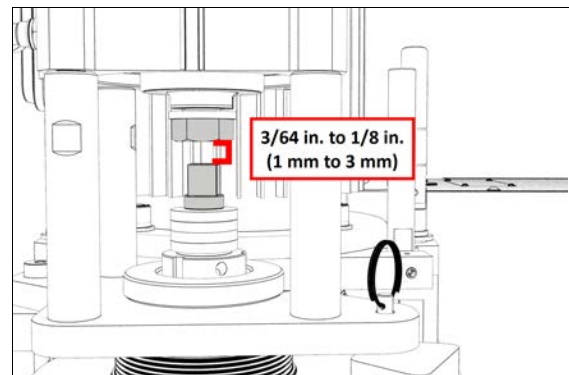


Figure 4-66: Correctly spaced drawbar and Power Drawbar cylinder.

2. Verify that the gap is between 3/64 in. and 1/8 in. (1 mm and 3 mm). Depending on the size of the gap, do one of the following:
 - **Between 3/64 in. and 1/8 in. (1 mm and 3 mm)**
You have completed adjusting the initial setup. Go to Operation.
 - **Less Than 3/64 in. (1 mm)** Go to Step 3.
3. Disconnect the shop's air supply from the Power Drawbar button.
4. Pull out the quick-release pin.

5. Pivot the Power Drawbar cylinder assembly to the left so that you can access the Power Drawbar cylinder's rod.

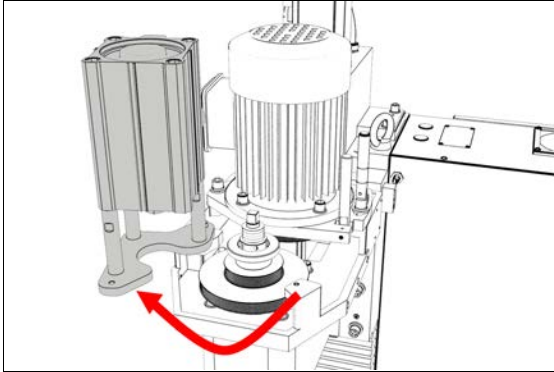


Figure 4-67: Power Drawbar cylinder pivoted to the left.

6. Remove the hex head screw on the Power Drawbar cylinder's rod with an adjustable wrench, and set it aside.

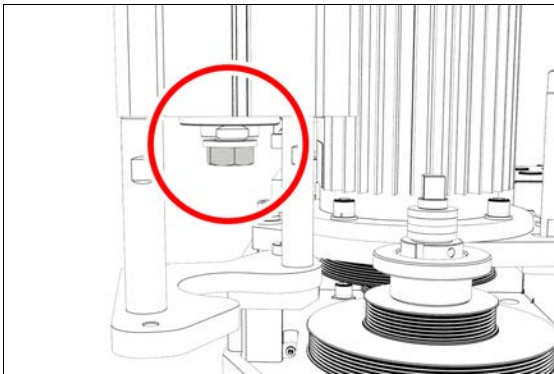


Figure 4-68: Hex head screw on the Power Drawbar cylinder's rod.

7. Remove the M16 washer from the Power Drawbar cylinder's rod, and set it aside.

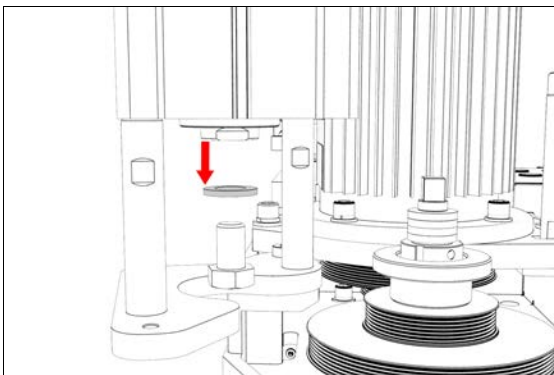


Figure 4-69: M16 washer removed from the Power Drawbar cylinder's rod.

8. Put the hex head screw back in, and then tighten it completely with an adjustable wrench.

9. Pivot the Power Drawbar cylinder to the original location.
10. Push in the quick-release pin.
11. Reconnect the shop's air supply to the Power Drawbar button.
12. Examine the space between the hex head screw on the Power Drawbar cylinder's rod and the top of the drawbar.
13. Verify that the gap is between 3/64 in. and 1/8 in. (1 mm and 3 mm). Depending on the size of the gap, do one of the following:
 - **Between 3/64 in. and 1/8 in. (1 mm and 3 mm)**
You have completed adjusting the initial setup.
 - **Less Than 1/8 in. (1 mm)** Go to Step 14.
14. Find the three provided M14 flat washers.
15. Put one M14 flat washer under each mounting post on the Power Drawbar cylinder.

4.7.2 Install the Flood Coolant Kit

Required Tools

- Adjustable wrench
- Center punch
- Clean rag
- Electric drill
- Isopropyl alcohol
- Marker
- Phillips screwdriver
- Silicone sealant
- Thread seal tape

Set up the Flood Coolant Kit

1. Put the coolant tank cover on top of the coolant tank.
2. Use four 10-32 x 1/2 in. screws and 10-32 nuts to mount the coolant pump to the coolant tank cover.

4: INSTALLATION

4.7 Install Accessory Components

- Put one end of the 3/4 in. ID clear tubing into the coolant tank cover. You can use a hose clamp on one end of the 3/4 in. ID clear tubing to prevent it from pulling out of the coolant tank.

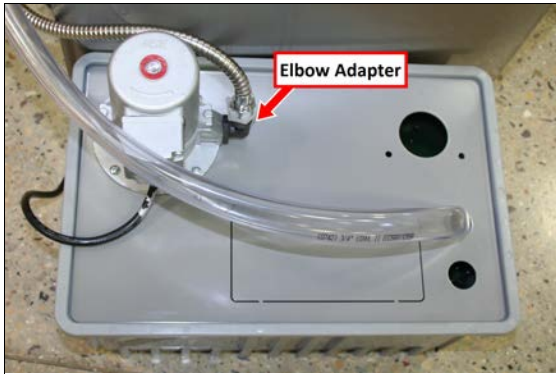


Figure 4-70: Coolant tank assembly.

- Wrap the threads of the coolant hose adapter with thread seal tape. Then, put it into the mounting bracket on the machine column.
- Tighten the fixing screw to secure the adapter in its position.
- Connect the 1/4 in. coolant hose to the coolant hose adapter.

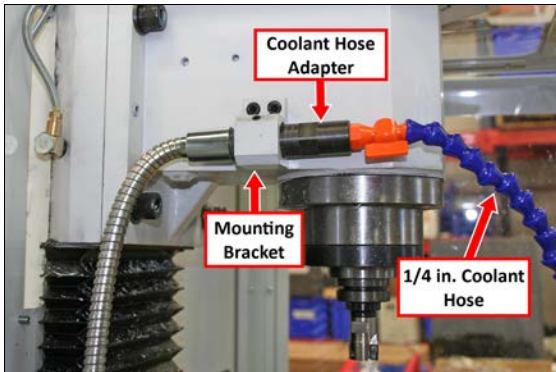


Figure 4-71: Coolant hose assembly.

- Put one provided silicone washer into each end of the chrome-armored hose.

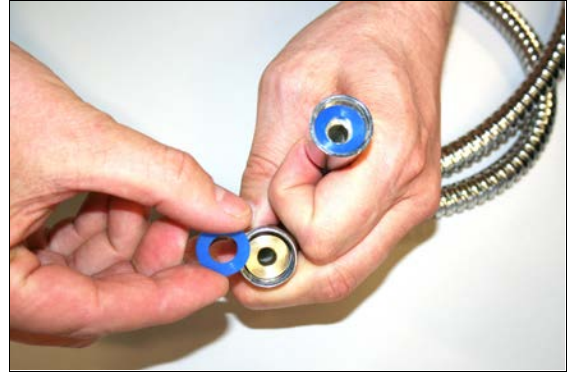


Figure 4-72: Silicone washers on the chrome-armored hose.

- Route the chrome-armored hose around the rear of the machine, and then connect it to the open end of the coolant hose adapter.
- Wrap the threads of the elbow adapter on the coolant pump with thread seal tape.
- Connect the loose end of the chrome-armored hose to the elbow adapter on the coolant pump.
- Remove the access panel from the chip pan.



Figure 4-73: Access panel on the chip pan.

- Put the provided chip screen in place of the chip pan's access panel.

13. Put one hose clamp on the loose end of the 3/4 in. ID clear tubing, and then attach it to the funnel tray.



Figure 4-74: 3/4 in. ID clear tubing attached to the funnel tray.

14. Use the funnel tray as a template: put it below the access panel on the chip pan, and then use a marker to mark its four holes.



Figure 4-75: The funnel tray below the access panel.

15. Use a center punch (or similar) to mark the four locations that you marked in Step 14. Then, use a 7/32 in. drill bit to drill the holes.
16. Use four 10-32 × 1/2 in. screws, rubber washers, and 10-32 nuts to attach the funnel tray to the chip pan.

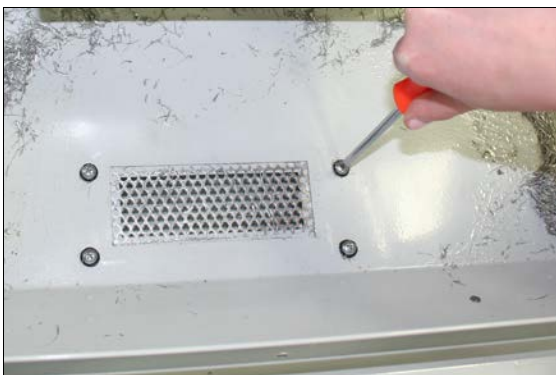


Figure 4-76: Funnel tray attached to the chip pan.

17. Use isopropyl alcohol and a rag to clean the perimeter of the funnel tray.
18. Apply a thin strip of silicone sealant around the perimeter of the funnel tray to seal against leaks.
19. Route the power cord on the coolant pump to the machine's power strip, and connect it to the power strip.

4.7.3 Install the PCNC 440 Automatic Tool Changer



IMPORTANT! Please read this section before you begin installing your Automatic Tool Changer (ATC).

- **PathPilot** You must update your controller to the latest version of PathPilot before installing and operating the ATC. If you don't, there's a risk that the ATC could become inoperable. Follow the instructions detailed in Before You Begin.

Required Tools

- 1-2-3 Block Set (PN 31950)
- 8 in. dowel for Tormach Tooling System (TTS) tool holder



Note: The dowel cannot be longer than 8 in.

- 1-1/2 in. adjustable wrench
- FRL Filter-Regulator-Lubricator (PN 38829)
- Metric hex wrench set
- Phillips screwdriver
- PCNC 440 Power Drawbar Kit (PN 35927)



Note: You must install the PCNC 440 Power Drawbar on your machine before installing the PCNC 440 Automatic Tool Changer (ATC). Do not install the power drawbar button — tool changes can be made on the ATC.

- Small, flat-head screwdriver
- Snips
- Tormach Tooling System (TTS) tool holder

Air Requirements

- **Air Pressure** 90-120 psi (620-825 kPa)
If the air supply is more than 120 psi (825 kPa), you must use a regulator.

4: INSTALLATION

4.7 Install Accessory Components

- **Air Volume** For successive, back-to-back tool changes, at least 2 cfm at 90 psi is required. Exact air volume depends on the frequency of tool changes.
- **Dry Air** We recommend using a compressed air dryer, desiccator, or filter between the air compressor and the machine.

Before You Begin

1. From the PathPilot interface, reference the mill. Select **Ref Z**, **Ref X**, and **Ref Y**.
2. Slowly jog the Z-axis down 1 1/2 in.



WARNING! Electrical Shock Hazard: You must power off the machine before making any electrical connections. If you don't, there's a risk of electrocution or shock.

3. Power off the machine and the PathPilot controller.
 - a. Push in the machine's red Emergency Stop button, which removes power to motion control.
 - b. From the PathPilot interface, select **Exit**.
 - c. Turn the Main Disconnect switch to **OFF** on the side of the electrical cabinet.

Prepare the Machine

1. Remove 10 socket head cap screws from the electrical cabinet with a 3 mm hex wrench, then set aside the screws and electrical cabinet cover.

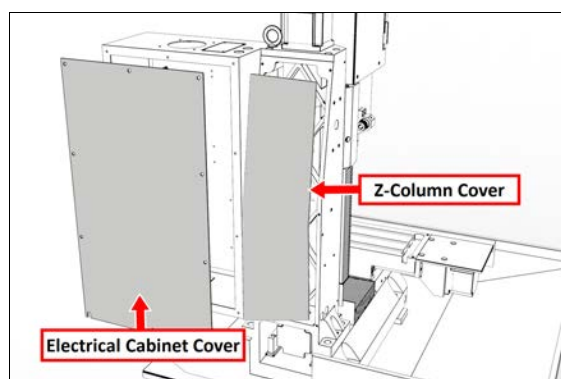


Figure 4-77: Components to remove from the machine.

2. Remove eight button head cap screws from the Z-column with a 3 mm hex wrench, then set aside the screws and the Z-column cover.

3. Remove four set screws from the side of the Z-column with a 4 mm hex wrench, then discard the screws.

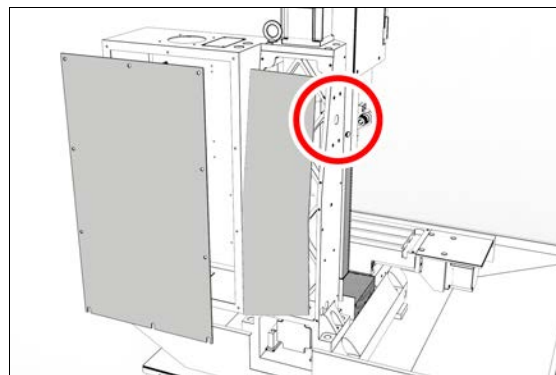


Figure 4-78: Set screw location on the Z-column.

Disassemble the Accessories

Before installing the Automatic Tool Changer (ATC), you must relocate, disassemble, or partially disassemble any of the following items, if you have them.

If you don't have any of these accessories, skip this section and go to Mount the Bracket.

- **Coolant Hose** If your machine has a coolant hose installed on the side of the spindle head, you must relocate it. Go to Disassemble the Coolant Hose.
- **Enclosure** If your machine has an enclosure, you must partially disassemble it. Go to Disassemble the Enclosure.
- **Power Drawbar Button** If your machine has a Power Drawbar button installed on the left side of the spindle head, you must disassemble it. Go to Disassemble the Button.

Disassemble the Coolant Hose

Use the provided kit to disassemble and relocate the coolant hose.

To disassemble the coolant hose:

1. Find and read the machine serial number. Depending on your specific machine, do one of the following:
 - **Serial Number 80075 and Below** Go to Step 2.
 - **Serial Number 80076 and Above** Go to Step 4.

2. Use the relocation bracket as a template: put it behind the spindle, flush with the right edge of the spindle head, mark the location of its holes with a marker.



Figure 4-79: Relocation bracket used as a template to mark the hole locations.

3. Drill and tap a hole for a 6 mm screw in the locations that you marked in Step 2.
4. Remove the coolant hose mount bracket and, if already installed, the 1/4-in. coolant hose from the side of the spindle head. Set both aside.

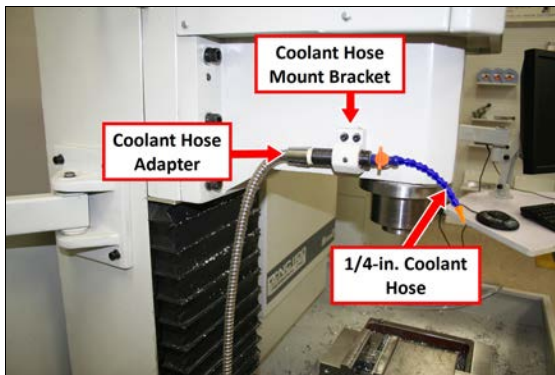


Figure 4-80: Flood coolant kit installed on the spindle head.

5. Attach the coolant hose mount bracket to the relocation bracket with the two screws from the coolant hose mount bracket.
6. Attach the relocation bracket below the spindle head.

Disassemble the Enclosure

1. Remove the nine 10-32 × 3/8 in. Phillips screws from the left rear panel with a Phillips screwdriver, then set aside the screws and the left rear panel.

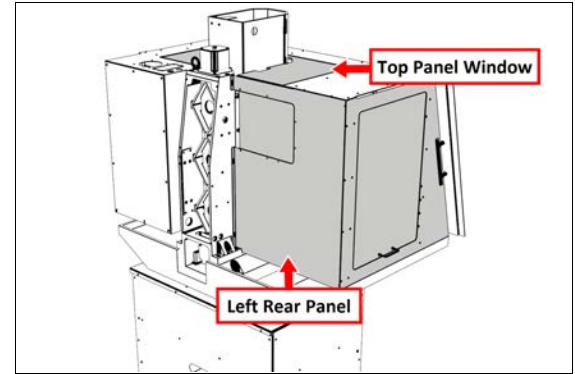


Figure 4-81: Components to remove on the enclosure.

2. Loosen the nine 10-32 × 3/8 in. Phillips screws on the top panel's three window retainers with a Phillips screwdriver.
3. Move the top window out of the window retainers, and then set it aside.

Disassemble the Power Drawbar Button

1. Disconnect the shop air supply from the machine.
2. Disconnect all air lines from the Power Drawbar button.
3. Remove one M6 × 12 mm socket head cap screw that secures the Power Drawbar button assembly's bracket to the spindle motor cover, and then set aside the M6 × 12 mm socket head cap screw.

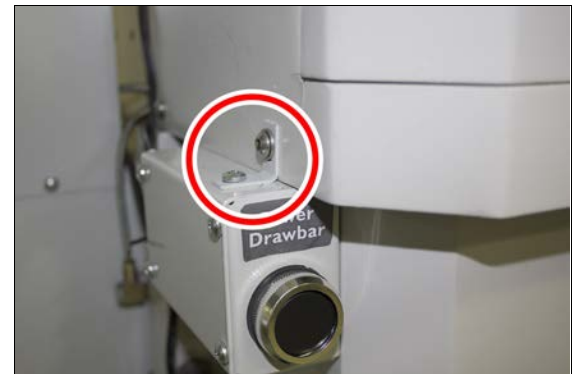


Figure 4-82: M6 × 12 mm socket head cap screw on the Power Drawbar button assembly.

4. Remove the Power Drawbar button from the spindle head.
5. Replace the M6 × 12 mm socket head cap screw on the spindle motor cover.

Mount the Automatic Tool Changer

1. Identify the four provided standoffs that are used to mount the ATC to the Z-column:

4: INSTALLATION

4.7 Install Accessory Components

- One fixed standoff, which uses a threadlocker to hold the nut in place
 - One tilt standoff
 - Two adjustable standoffs
2. Remove the flange nuts and the washers from the standoffs, and then set them aside.
 3. Arrange the four standoffs next to each other as shown in the following image.

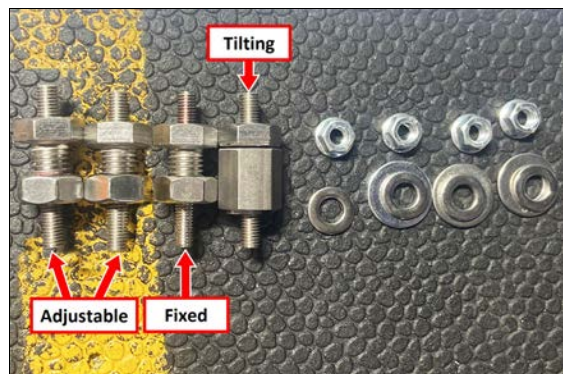


Figure 4-83: Four standoffs arranged next to each other.

4. Loosen or tighten the nut on the two adjustable standoffs until they're the same length as the fixed standoff.
5. Install the four standoffs on the Z-column as shown in the following image.

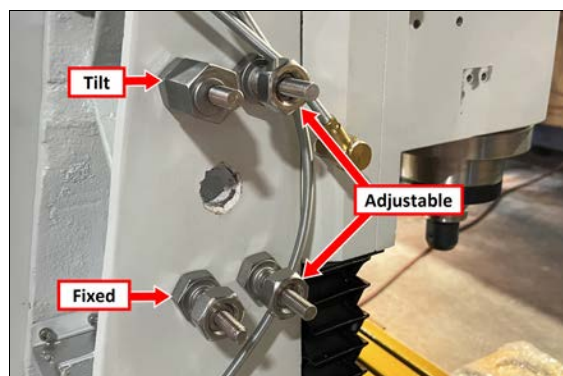


Figure 4-84: Four standoffs installed on the Z-column.

6. Securely tighten the standoffs on the Z-column with a 1-1/2 in. adjustable wrench.
7. Remove four M4 × 6 mm screws that secure the access plate to the ATC motor enclosure with a 2.5 mm hex wrench, and then set aside the screws and the access plate.

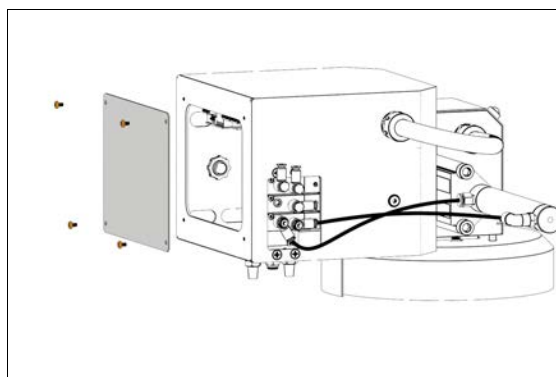


Figure 4-85: ATC motor enclosure with access plate removed.

8. Hold the oil line above the four standoffs, and then put all ATC wires through the Z-column's access slot.

CAUTION! Team Lift Required: You must have the aid of more than one person to lift and move the object. The object is heavy, and lifting it by yourself can cause serious injury.

9. While continuing to hold the oil line above the standoffs, lift the ATC, and then temporarily rest it on the standoffs as shown in the following image.

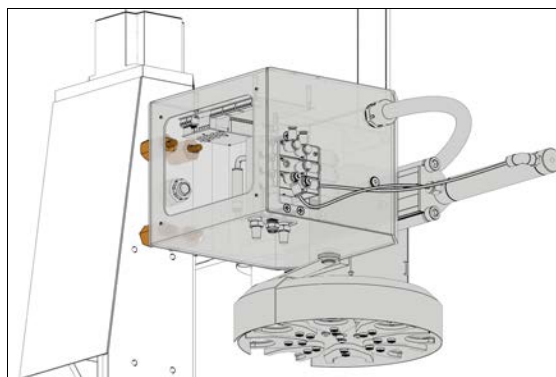


Figure 4-86: ATC rested on the standoffs.

10. Release the oil line.
11. Secure the bracket by reinstalling the washers and flange nuts that you removed in Step 2 with a 13 mm socket.

12. Identify the two pre-installed bolts on the end of the ATC's linear rail. Remove them and set them aside.



Figure 4-87: Bolts for the air cylinder pre-installed on the ends of the ATC's linear rail.

13. Identify the air cylinder that's included in this kit. Insert it into the slot on the linear rails as shown in the following image.



Figure 4-88: Installing the air cylinder into the ATC.

14. Secure the air cylinder to the ATC with the bolts that you set aside in Step 12 and an 8 mm hex wrench.
15. Make the air line connections as follows:
 - a. Connect the short air line from the left port of the lower solenoid valve to the fitting on the air cylinder that's closest to the spindle.
 - b. Connect the long air line from the right port of the lower solenoid valve to the fitting on the air cylinder that's furthest from the spindle.

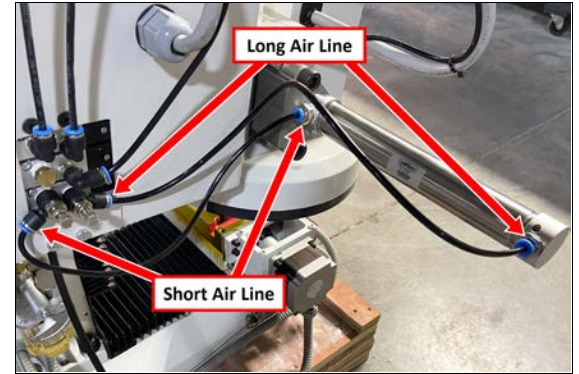


Figure 4-89: Air line connections.

Make Air Connections

1. Manually move the tool tray toward the spindle.
2. Identify the top and bottom solenoids on the left side of the motor enclosure.

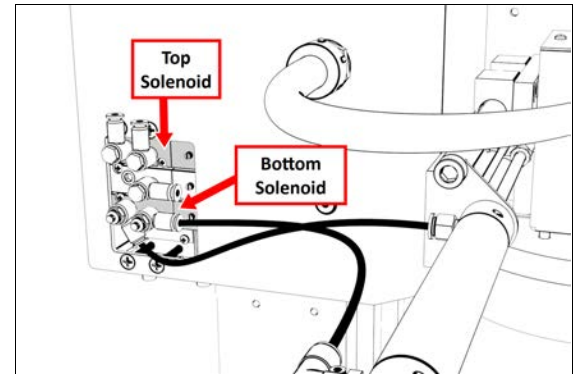


Figure 4-90: Solenoids.

3. Push in the screw on the bottom solenoid with a small, flat-blade screwdriver, and then turn it clockwise. The screw locks in place to hold the ATC in the tray load position.
4. From the power drawbar cylinder, identify two 5/32 in. plastic tubes.



WARNING! Crush Hazard: If the ATC isn't completely retracted, it could move once the air is reconnected. When you reconnect the air, you must keep your hands away from the ATC.

4: INSTALLATION

4.7 Install Accessory Components

5. the loose ends of the 5/32 in. plastic tubes in the following order:
 - a. Connect the Retract airline (from to the bottom push-to-connect elbow on the power drawbar cylinder) to the left push-to-connect elbow on the ATC's top solenoid.
 - b. Connect the Advance airline (from the T-connector on the power drawbar cylinder) to the right push-to-connect elbow on the ATC's top solenoid.
 - c. Connect the air supply line from the FRL to the air in port in the ATC main assembly, as shown in the following image.

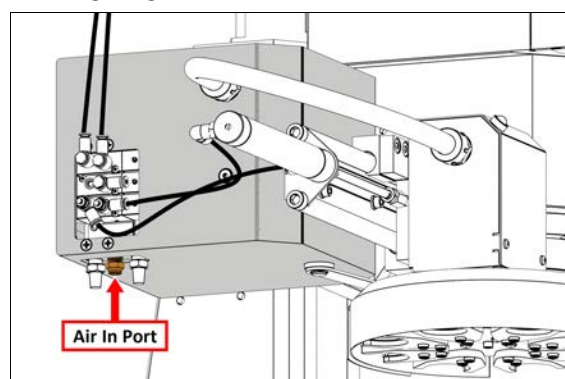


Figure 4-91: Air in port in the ATC main assembly.

6. Depending on your air supply accessories, do one of the following:
 - **(Optional) FRL Filter-Regulator-Lubricator** Go to (Optional) Install the .
 - **No FRL** Power on the air compressor, and then go to Level the .

(Optional) Install the FRL Filter-Regulator-Lubricator

The FRL Filter-Regulator-Lubricator (PN 38829) is an optional accessory for the Automatic Tool Changer (ATC) that lubricates the air with air tool oil. For information on installation and use, see the documentation that ships with the FRL Filter-Regulator-Lubricator.

Level the Automatic Tool Changer

This section details the procedure required to roughly align the ATC on the mill. More adjustments are made later in this document.

1. Make sure that the ATC's carousel door opening is approximately equal in distance on all sides of the spindle. If adjustments are needed:

- **Forward or backward:** Pull or push the ATC until it's centered.
- **Right or left:** Use an adjustable wrench to loosen or tighten the nut on the adjustable standoffs (located between the motor box and the Z-column), and move the ATC until it's centered.

2. Make sure that the motor enclosure is approximately 1/8 inch from the mill's spindle head. If adjustments are needed:
 - a. Use an adjustable wrench to loosen or tighten the nut on the adjustable standoffs.
 - b. Move the ATC assembly right or left until it's at the correct distance.

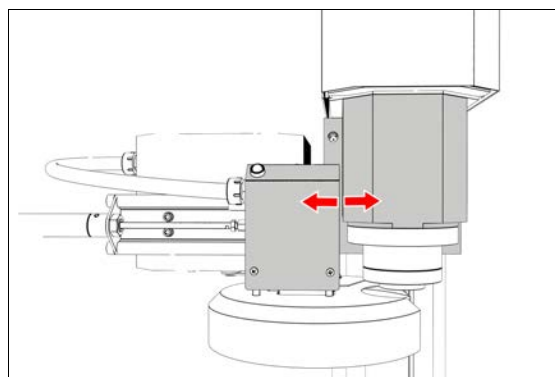


Figure 4-92: ATC's motor enclosure distance from mill's spindle head.

3. Make sure that the front side of the motor enclosure is approximately parallel with the mill's spindle head. If adjustments are needed, use an adjustable wrench to turn the nut on the tilt standoff until it's parallel.

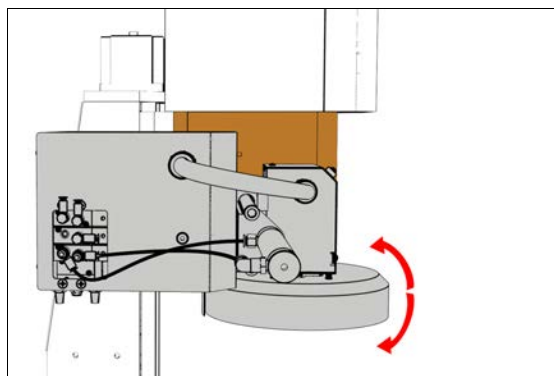


Figure 4-93: Adjusting the ATC's motor enclosure to parallel.

4. From the four standoffs, securely tighten four flange nuts.

5. From the left side of the motor enclosure, on the bottom solenoid, use a small flat-head screwdriver to turn the screw on counterclockwise.
The tool tray retracts.

Make Electrical Connections

1. Put wire V+ and V- (red and black power wires) through the Z-column toward the power supply.
2. Make the following power connections on the power supply with a Phillips screwdriver:
 - a. Connect wire V+ (red) to V+
 - b. Connect wire V- (black) to V-
3. Put wire VFD-1 and VFD-8 (yellow and white spindle wires) through the access hole in the bottom of the Z-column toward the electrical cabinet.
4. Remove the left-hand, bottom, and middle wire trough covers from the electrical cabinet.
5. Put wire VFD-1 and VFD-8 (yellow and white spindle wires) through the wire troughs together toward the variable frequency drive (VFD).
6. Make the following spindle connections on the VFD with a small, flat-head screwdriver:
 - a. Connect wire VFD-1 (yellow) to pin 1 of the upper connector (+5v)
 - b. Connect wire VFD-8 (white) to pin 2 of the lower connector (HA)



Note: Both pins already have wires connected, so you'll need to piggyback VFD-1 and VFD-8 onto an already populated connector.

7. Replace the wire trough covers inside the electrical cabinet.
8. Put the electrical cabinet cover back on the electrical cabinet with the 10 socket head cap screws set aside in Prepare the Machine.
9. Route the USB cable toward the rear of the machine.
10. Put the USB cable in the access hole in the rear of the stand toward the controller.
11. Put the USB cable into an open USB port on the PathPilot controller.

IMPORTANT! Do not connect the ATC USB cable to a USB hub. Make the connection directly to the PathPilot controller itself. If you don't have any free USB ports on the controller, connect a different accessory to a USB hub.



12. Put the Z-column cover back on the rear of the machine with the eight button head cap screws set aside in Prepare the Machine
13. Put the ATC motor enclosure's access plate back with the four M4 × 6 mm screws set aside in Mount the .

Verify the Installation

1. Power on the machine and the PathPilot controller.
 - a. Turn the PathPilot controller strip to **ON**.
 - b. Turn the Main Disconnect switch to **ON** on the side of the electrical cabinet.
 - c. Twist out the machine's red Emergency Stop button, which enables movement to the machine axes and the spindle.
 - d. Press the Start button.
 - e. Bring the machine out of reset and reference it.

IMPORTANT! You must update your controller to the latest version of PathPilot before operating the Automatic Tool Changer. If you don't, there's a risk that the ATC could become inoperable.



2. If you have not yet done so, you must make sure that the PathPilot controller is updated to the latest version of PathPilot: From the **Status** tab, select **Update**. Then, follow the on-screen instructions.

4: INSTALLATION

4.7 Install Accessory Components

- From the PathPilot interface, on the **Settings** tab, select the **ATC** radio button.

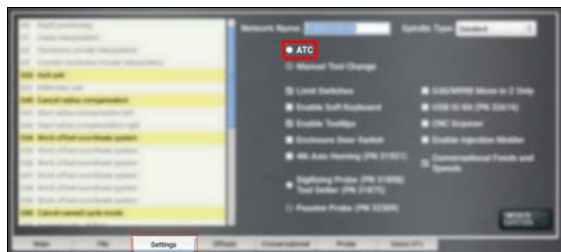


Figure 4-94: ATC radio button on the Settings tab.

The **ATC** tab appears in the PathPilot interface.



Note: If prompted, you may need to update the firmware for the ATC. Follow the on-screen instructions.

- Load a tool into the spindle:
 - Push and hold the collet open button on the side of the ATC.



Figure 4-95: Collet open button on the side of the ATC.

The collet opens.

- Load a tool into the spindle.
 - Release the button.
The collet closes.
- From the PathPilot interface, on the **Main** tab, in the **RPM DRO** field, type 1000. Then select the **Enter** key.
 - Select **FWD**.
The spindle starts.
 - From the **Status** tab, make sure that the **VFD Running** green light comes on.



Figure 4-96: VFD Running light on the Status tab.

- Select **Stop**.
The spindle stops.
- If the **VFD Running** light didn't come on in Step 7:
 - Use the button on the ATC to remove the tool from the spindle.
 - Power off the machine and the PathPilot controller.
 - Push in the machine's red Emergency Stop button, which removes power to motion control.
 - From the PathPilot interface, select **Exit**.
 - Turn the Main Disconnect switch to **OFF** on the side of the electrical cabinet.
 - Examine the wire connections on the VFD. Make sure that both wires are properly connected, and then repeat the steps in this section.
- Use the button on the ATC to unload the tool from the spindle.

Make Final Alignments to the Automatic Tool Changer

Adjust the Tool Tray and Slots

1. Make sure there is no tool in the spindle.
2. From the PathPilot interface, on the **ATC** tab, select **Ref Tool Tray**.

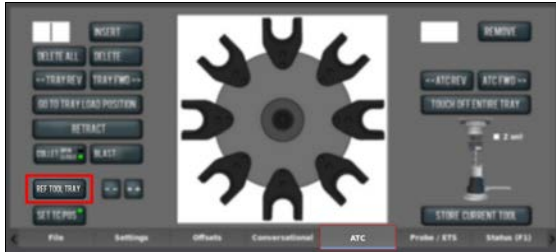


Figure 4-97: Ref Tool Tray button.

The tool tray spins.

Note: You are only required to reference the tool tray once — unlike the mill axes' referencing procedure.

3. Select **Go to Tray Load Position**.

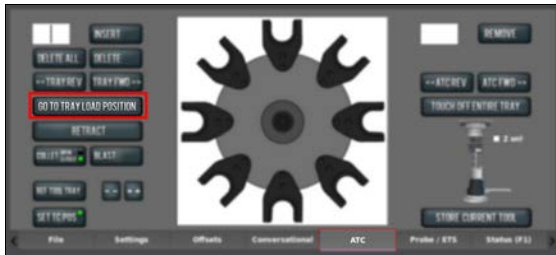


Figure 4-98: Go to Tray Load Position button.

4. When prompted, select **OK**.
The tool tray moves forward.
5. Make sure the spindle is centered over the ATC's fork.

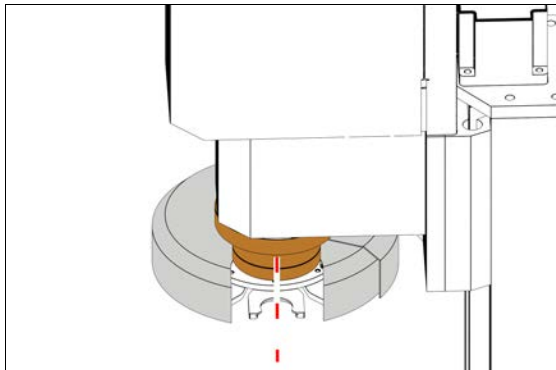


Figure 4-99: Spindle centered over the fork.

Adjust the Tool Tray Load Position

If the spindle is not centered over the fork in the X direction, you must manually adjust the position of the ATC assembly on the linear rails.

1. From the ATC's linear rail, loosen the jam nut.

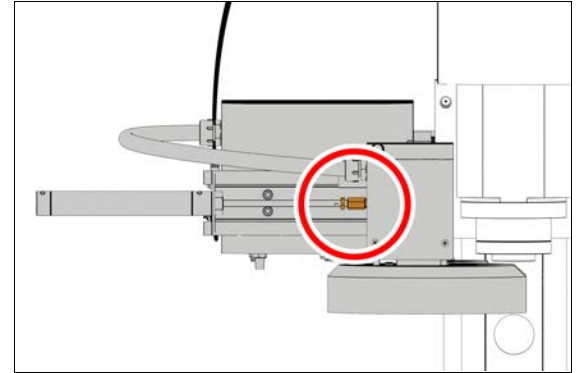


Figure 4-100: Jam nut on the linear rail.

2. Slide the coupler to bring the ATC assembly closer to the spindle head or further away from the spindle head.
3. After the ATC's carousel door opening is positioned correctly with the spindle, re-tighten the jam nut.

Adjust the Tilt of the Automatic Tool Changer

1. Insert a dowel — no more than 8 inches long — into a TTS tool holder.
2. Insert the tool holder into the fork. Make sure the groove in the tool holder slides along the shoulder on the fork.

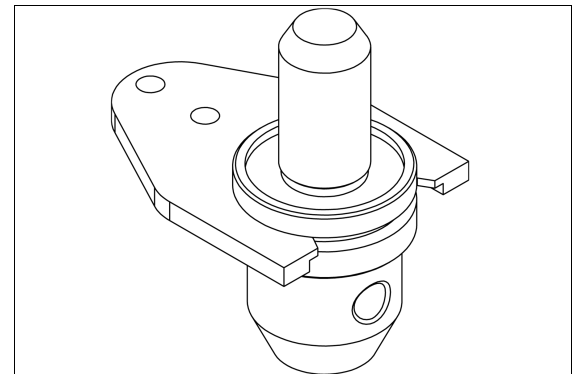


Figure 4-101: TTS tool in an ATC fork.

Note: Do not rest the tool holder on top of the fork.

4: INSTALLATION

4.7 Install Accessory Components

3. From the PathPilot interface, on the **ATC** tab, type 1 in the **Tool Number DRO** field.



Figure 4-102: Tool Number DRO field.

4. Select **Insert**.
The tool number entered appears on the diagram of the ATC's carousel.
5. Place a 1-2-3 block on the machine table. Make sure the dowel is parallel with the 1-2-3 block in both the Y and X direction.

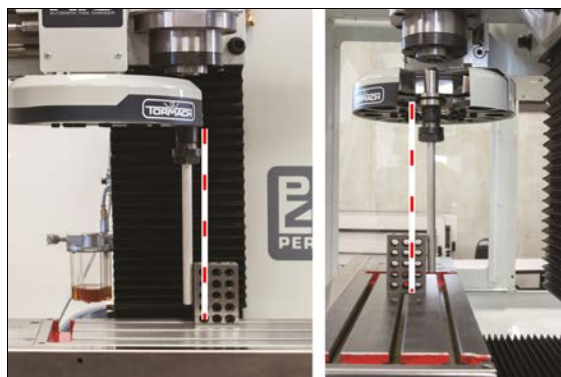


Figure 4-103: Dowel parallel in both the Y and X direction.

6. If the dowel is not parallel, do one of the following:
 - From the ATC's linear rail, loosen two 5 mm socket head cap screws and slowly pivot the ATC assembly until the dowel is parallel with the 1-2-3 block; re-tighten 5 mm socket head cap screws. Then, re-check the alignment of the dowel in both the Y and X direction.

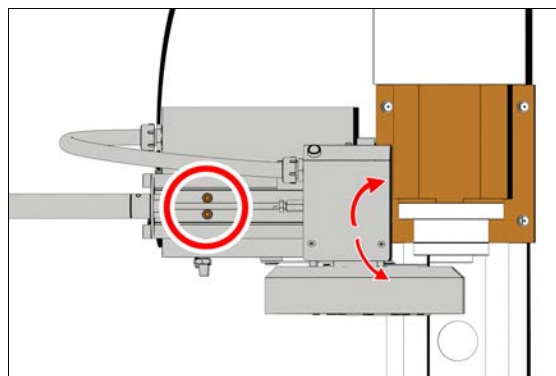


Figure 4-104: Pivoting the ATC using the linear socket head cap screws.

- From the tilt standoff, use an adjustable wrench to turn the nut and slowly pivot the ATC assembly until the dowel is parallel with the 1-2-3 block. Then, re-check the alignment of dowel in both the Y and X direction.

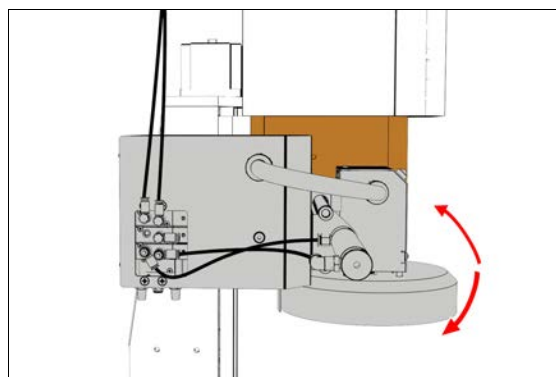


Figure 4-105: Pivoting the ATC using the tilt standoff.

7. Repeat Step 6 until the dowel is parallel with the 1-2-3 block.
8. Remove the 1-2-3 block from the machine table.

Adjust the Tool Tray Rotation

1. From the PathPilot interface, slowly jog the Z-axis down to bring the spindle nose toward the tool in the ATC's tool tray.
2. Make sure the tool's shank is in line with the collet in the spindle.

3. Examine the alignment of the spindle's centerline to the tool slot's centerline. Determine if the tray must move clockwise or counterclockwise.

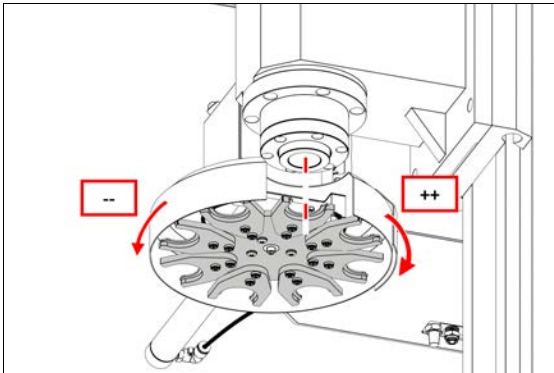


Figure 4-106: Spindle centerline aligned with tool slot centerline.

4. From the PathPilot interface, in the **ATC** tab, do one of the following:
 - Select -- to step the tool tray counterclockwise.
 - Select ++ to step the tool tray clockwise.



Figure 4-107: Buttons used to rotate the tool tray.

Validate the Automatic Tool Changer Alignment

1. From the PathPilot interface, slowly jog the Z-axis down over the tool's shank.
2. Make sure the tool's shank moves freely in the collet. If it does not, this indicates that the ATC is misaligned and you must repeat the steps in the "Adjust the Tool Tray and Slots" (page 69).

Setting the Tool Change Height

1. From the PathPilot interface, slowly jog the Z-axis down over the tool; stop jogging when the spindle nose just makes contact with the shoulder of the tool holder.
2. On the **ATC** tab, click the **Set TC POS** button. The tool change position is set.

Adjusting for Rotational Play

There is a small amount of rotational play built into the ATC carousel. This play allows for some misalignment during tool changes, and must be adjusted for in both directions. The taper on the tool shank also helps align the tool during a tool change.

1. From the PathPilot interface, on the **ATC** tab, click the **Tray FWD** button to rotate the tray clockwise (forward) one full tool slot.
2. Click the **Tray REV** button to rotate the tray counterclockwise (backward) one full tool slot.
3. Make sure the tool's shank is in line with the collet in the spindle. If it is not, "Adjust the Tool Tray Rotation" (on the previous page).

Re-Assembly

1. From the enclosure's left back panel, remove the ATC cover.
2. Use nine 10-32 x 3/8 in. Phillips screws set aside earlier in this procedure to re-install the enclosure's left back panel to both the machine column and the enclosure's left side panel.
3. Slide the top window back in place on the enclosure's top panel and tighten nine 10-32 x 3/8 in. Phillips screws.

4.7.4 Install the Enclosure

Before You Begin

1. Inspect the item(s):
 - Photograph any damage that may have occurred during shipping.
 - Verify the received goods against the packing list.

If there is any damage or shortages, you must contact Tormach within 30 days of receipt. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.
2. Read the packing list and locate all items.



Tip! Keep all screws one quarter-turn loose while installing the enclosure panels. This makes it easier to align the panels. Once you're done installing the enclosure, fully tighten all screws.

4: INSTALLATION

4.7 Install Accessory Components

Required Tools

- Adjustable wrench
- Electric drill
- Flat-blade screwdriver, small
- Phillips screwdriver
- Tap handle

Install the Enclosure Panels

Tip! To install the enclosure panels, **work from the left to the right of the machine.**

To make it easier to install all components, **don't install the side windows until later in the procedure.** **Keep all screws one quarter-turn loose while installing the enclosure panels.** This makes it easier to align the panels. Once you're done installing the enclosure, fully tighten all screws.



Install the Left Rear Panel

1. Loosen the four bolts that secure the machine to the chip pan. This allows for correct positioning of the enclosure.
2. Put a strip of butyl tape on the top of the chip pan in the locations shown in the following image.

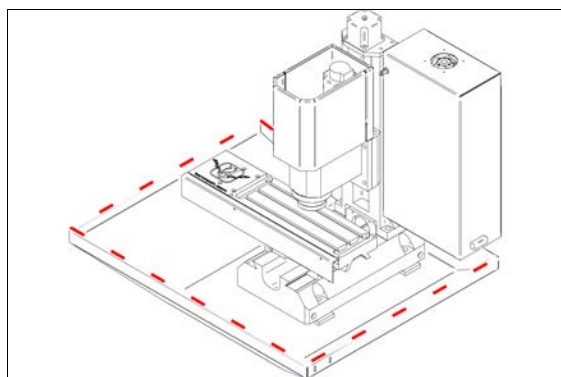


Figure 4-108: Locations for butyl tape on the top of the chip pan.

3. Attach the ATC cover to the opening on the inside of the left rear panel with seven M5 × 0.8 - 10 screws. The screw heads must face the inside of the left back panel when it's installed.

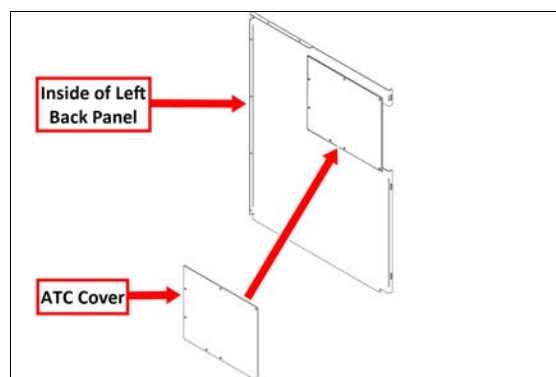


Figure 4-109: ATC cover attached to the opening on the inside of the left rear panel.

NOTICE! If the screw heads aren't facing inside, the screws could interfere with the oil line routing.

4. **Serial Numbers 80003-80075 Only** If your machine's serial number is between 80003 and 80075, you must drill and tap three holes on the machine column before you install the left rear panel. Complete the following steps:
 - a. Use the left back panel as a template: put it on the left rear side of the chip pan, and mark the three holes on the machine column.
 - b. Set the left rear panel aside.
 - c. Make three hole locations in the machine column with the #21 drill and M5 tap (both provided).
5. Attach the left rear panel to the machine column with three M5 × 0.8 - 10 screws.

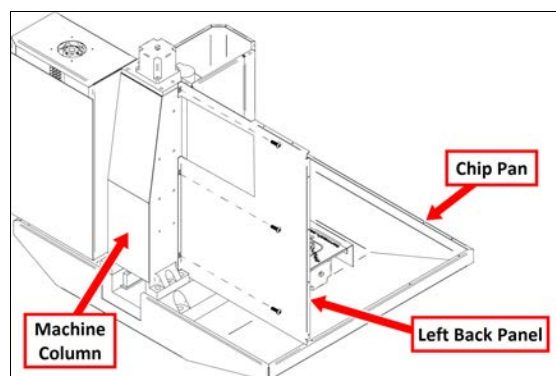


Figure 4-110: Left rear panel attached to the machine column.

Install the Left Side Panel

1. Attach the left side panel to the left rear panel with four M5 × 0.8 - 10 screws.

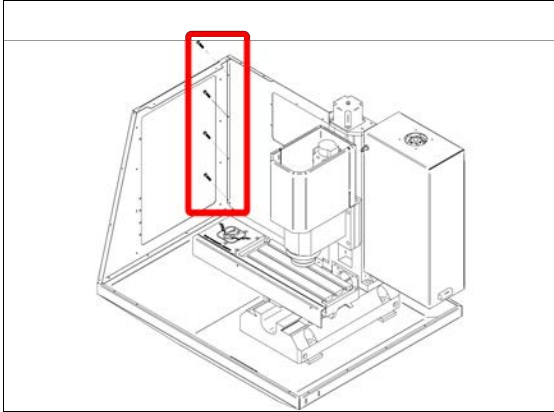


Figure 4-111: Left side panel attached to the left rear panel.

2. Attach the left side panel to the left chip pan with four M5 × 0.8 - 10 screws.

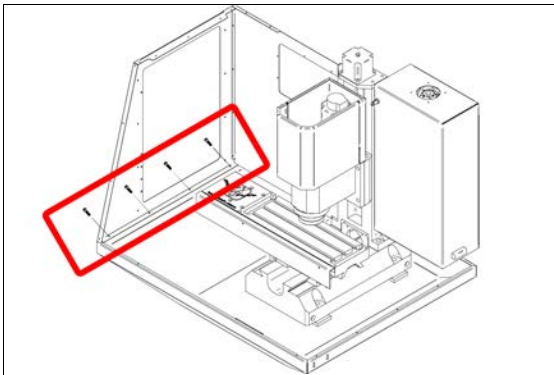


Figure 4-112: Left side panel attached to the left chip pan.

Install the Left Side Window Retainers



Note: To make it easier to install all components, don't install the side windows yet. Wait until later in the procedure.

1. Loosely install two vertical window retainers on both sides of the left side panel's window opening with two sets of four M5 × 0.8 - 10 screws.
Don't fully tighten the screws. Leaving the window retainers loose makes it easier to install the side windows.

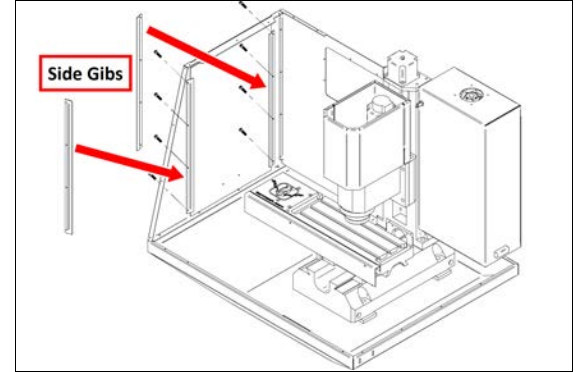
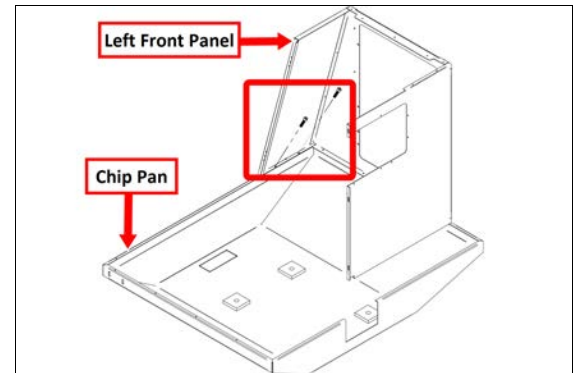


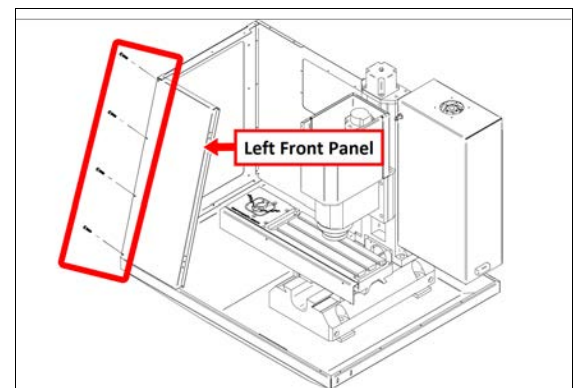
Figure 4-113: Left side window retainers attached to the left side panel.

Install the Left Front Panel

1. Loosely attach two M5 × 0.8 - 10 screws to the chip pan.
2. Slide the left front panel over the two screws that you installed in Step 1.



3. Attach the left front panel to the left side panel with four M5 × 0.8 - 10 screws.

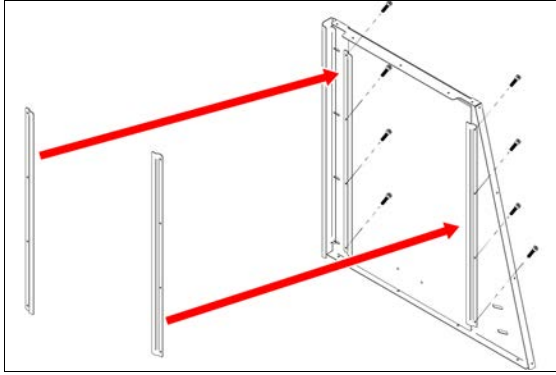


Install the Right Side Panel

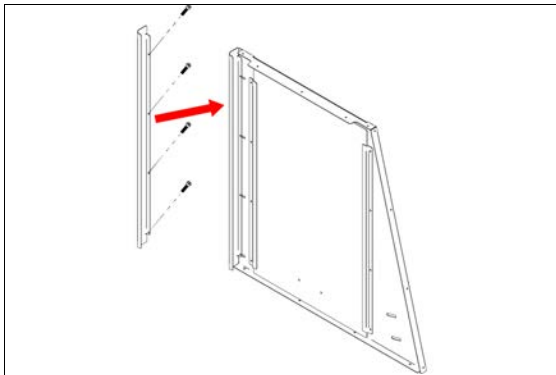
1. Loosely install two vertical window retainers on both sides of the right side panel's window opening with two sets of four M5 × 0.8 - 10 screws.

4: INSTALLATION

4.7 Install Accessory Components



2. Attach the right side support to the right side panel with four M5 × 0.8 - 10 screws.



3. Attach the right side panel to the right chip pan with four M5 × 0.8 - 10 screws.

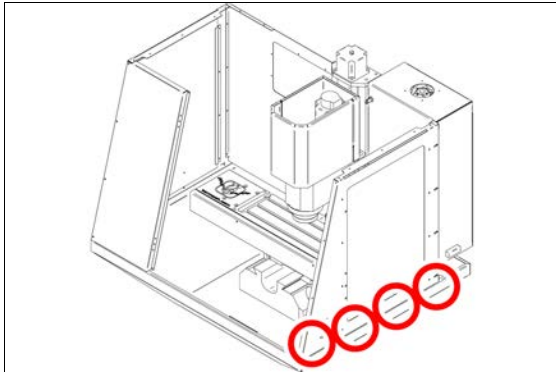
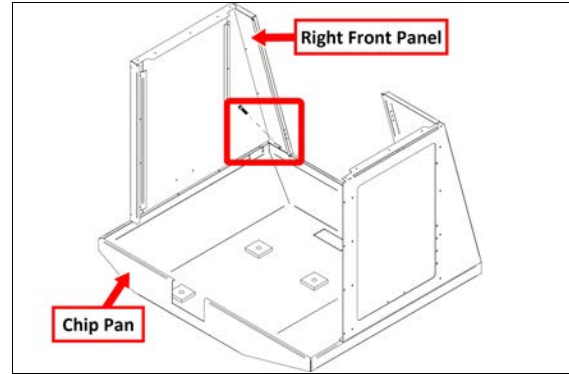


Figure 4-114: Right side panel attached to the chip pan.

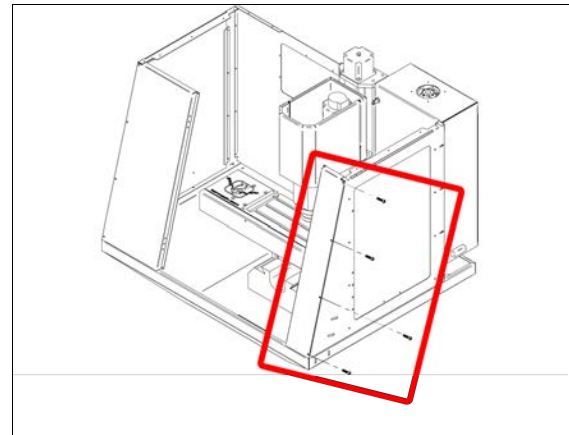
4. (Optional) If required, pull the chip pan forward while pushing the machine backward until you can secure the screws in place.

Install the Right Front Panel

1. Loosely attach one M5 × 0.8 - 10 screw to the chip pan.
2. Slide the right front panel over the screw that you installed in Step 1.



3. Attach the right front panel to the right side panel with four M5 × 0.8 - 10 screws.



Install the Top Panel

1. Loosely attach two short, vertical window retainers to both sides of the top panel with three sets of M5 × 0.8 - 10 screws.
2. Loosely attach the long, vertical window retainer to the top panel with three M5 × 0.8 - 10 screws.

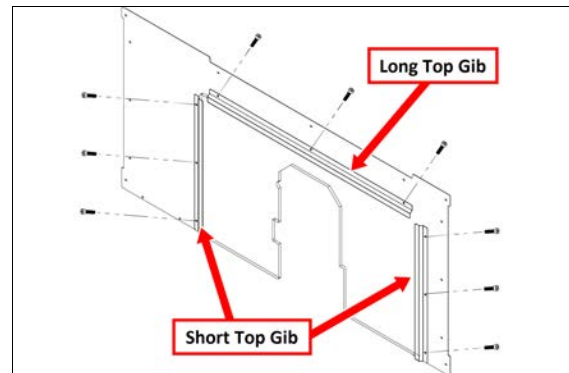


Figure 4-115: Window retainers attached to the top panel.

3. Remove and discard the protective plastic film from the window.

4. Slide the window into the three installed window retainers. Take care not to scratch the window.
5. Attach the top panel assembly to the enclosure with 12 M5 × 0.8 - 10 screws, as shown in the following image.

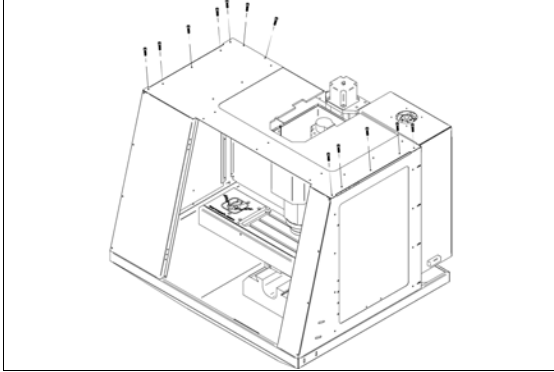


Figure 4-116: Top panel attached to the enclosure.

Install the Top and Bottom Trim

1. Attach the top trim to the right and left front panels with two M5 × 0.8 - 10 screws, two M5 flat washers, and two M5 nuts.

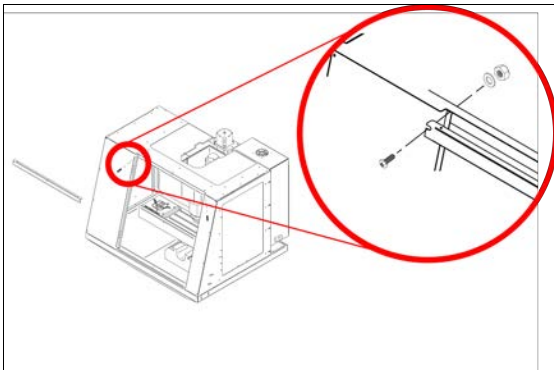


Figure 4-117: Attaching the top trim to the right and left front panels with M5 screws, washers, and nuts.

2. Attach the top trim to the top panel with three M5 × 0.8 - 10 screws.

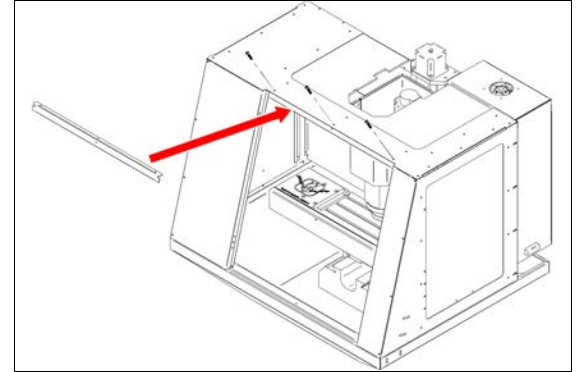


Figure 4-118: Attaching the top trim to the top panel with M5 screws.

3. Attach the bottom trim to the right and left front panels with two M5 × 0.8 - 10 screws, two M5 flat washers, and two M5 nuts.

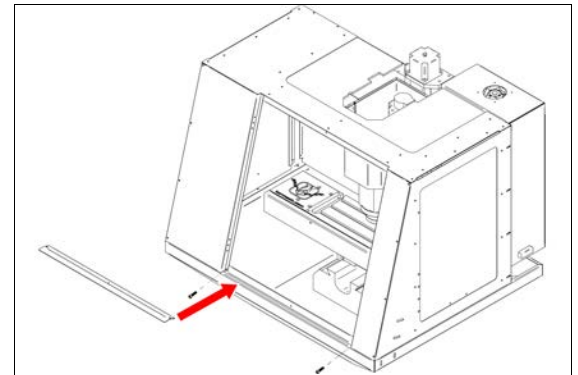
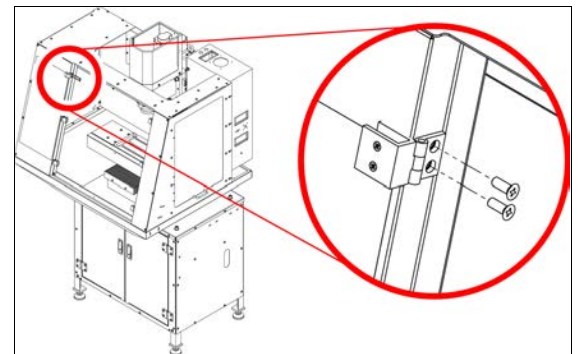


Figure 4-119: Attaching the bottom trim to the right and left front panels with M5 screws, washers, and nuts.

Install the Front Doors

Assemble the Front Doors

1. Install three door hinges onto either side of the enclosure's door opening. Install each door hinge with two M5 × 0.8 - 14 Phillips screws and a Phillips screwdriver.



4: INSTALLATION

4.7 Install Accessory Components

Figure 4-120: Installing the door hinges.

2. Insert the left door window into the hinges on the left side of the opening. Keep the two holes on the edge of the door toward the center of the enclosure — they're used to mount the door handle.

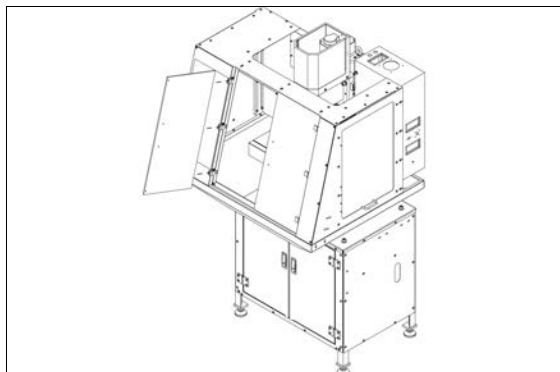


Figure 4-121: Installing the left door window onto the hinges.

3. Repeat Step 2 to install the right door.
4. Find the door edge panel, and install it onto the left door with a 2.5 mm hex wrench and 7 mm wrench.

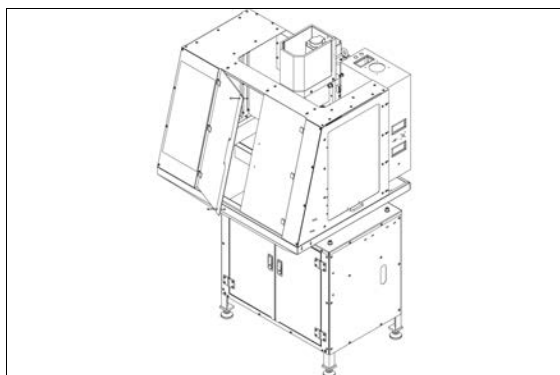


Figure 4-122: Installing the door edge panel onto the left door.

5. Find the two door handles. Install one onto each door with two M4 × 0.7 - 16 button head cap screws and a 2.5 mm hex wrench.

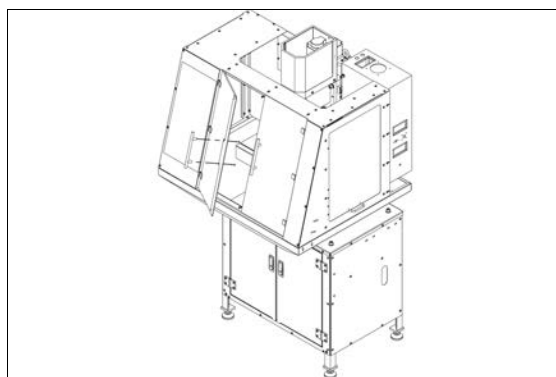


Figure 4-123: Installing the door handle.

Install the Front Door Assemblies

1. Attach the door hinges on each door windows to the enclosure in the orientation that you determined in Assemble the Front Doors with 12 (missing or bad snippet) flat head Phillips screws and 12 10-32 nuts.
2. Attach one large handle to the right door window with two 8-32 × 1/2-inch Phillips screws.

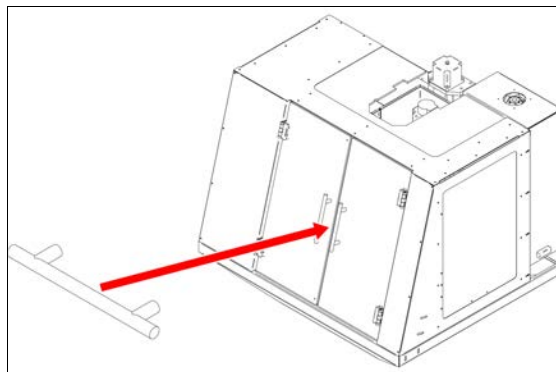


Figure 4-124: Large door handle attached to the right door window.

Install the Side Windows

1. Remove and discard the protective plastic film from the windows.
2. Slowly slide one window through the left side panel's window opening and into the two installed vertical window retainers. Take care to not to scratch the window.
3. Attach one small handle to the left side window with two 8-32 × 1/2 in. Phillips screws.

- Repeat Steps 2 to 3 for the right side window.

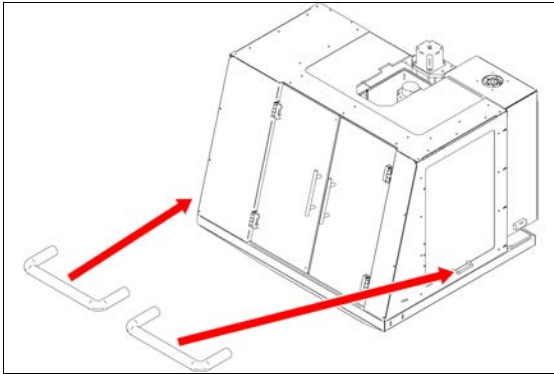


Figure 4-125: Small handles attached to the side windows.

- Securely tighten the four bolts securing the machine to the chip pan.
- To more easily install the enclosure components, you didn't fully tighten various screws in previous installation steps. At this point, securely tighten all screws used in the procedure.
You have completed assembling and installing the PCNC 440 Enclosure.

4.7.5 Door Switch Installation

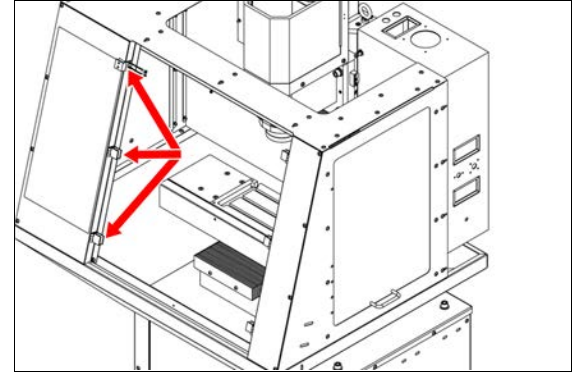
Required Tools

- 1/4-in. drill bit
- 2 mm hex wrench
- 3 mm hex wrench
- 5/16-in. box wrench
- Electric drill
- Phillips screwdriver
- Small, flat-blade screwdriver

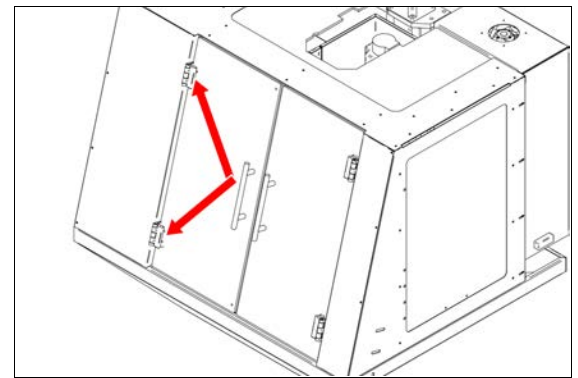
Install the Door Switches

The installation of the door switch kit varies depending on the PCNC 440 enclosure version. The quickest way to determine which enclosure you're using is by looking at the hinges on the front of the doors:

- PCNC 440 Enclosure Kit (PN 51469)** The newer version of the enclosure kit has three hinges per door. Go to "3-Hinge Doors (Newer Enclosure)" (below).



- PCNC 440 Enclosure Kit (PN 35605)** The older version of the enclosure kit has two hinges per door. Go to "2-Hinge Doors (Older Enclosure)" (on the next page).



3-Hinge Doors (Newer Enclosure)

- Power off the machine and the PathPilot controller.
 - Push in the machine's red Emergency Stop button, which removes power to motion control.
 - From the PathPilot interface, select **Exit**.
 - Turn the Main Disconnect switch to **OFF** on the side of the electrical cabinet.
- Identify the orientation of the door switches:
 - The door switch with the **shorter** wires is installed on the right side of the enclosure.
 - The door switch with the **longer** wires is installed on the left side of the enclosure.
- Open both doors on the enclosure.
- Mount the door switch bracket onto the screws for the top door hinge on the inside of the enclosure.

4: INSTALLATION

4.7 Install Accessory Components

5. Use the two provided M5 nuts to secure each door switch to the top door hinge.

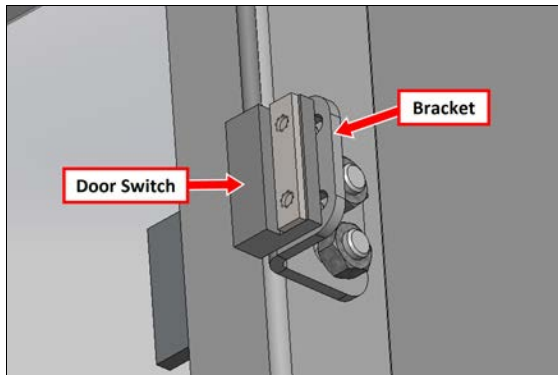


Figure 4-126: Example of the door switch installed as viewed from the inside of the enclosure.

6. Use a Phillips screwdriver to loosen the set screws on each top door hinge.
7. Slide a magnet bracket between each door window and top door hinge.

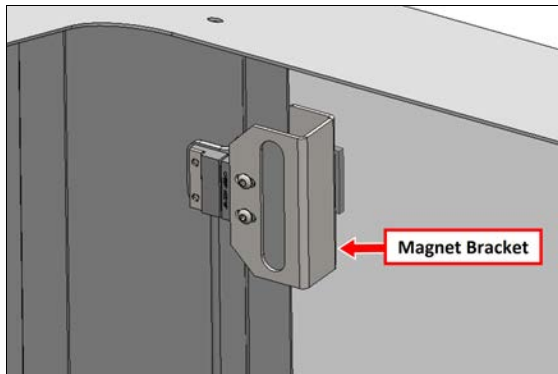


Figure 4-127: Example of an installed magnet bracket.

8. Tighten the screws on each door hinge to secure the magnet brackets to the door windows.

NOTICE! To prevent window damage from over-tightening, use your fingers to tighten the screws.

9. Close each door on the enclosure, and make sure that the magnet contacts the door switch. If needed, loosen the screws on the switch and magnet and adjust the fitment of the door switch and magnet within the slot.
10. Identify two M5 button head cap screws on the enclosure as shown in the following image.

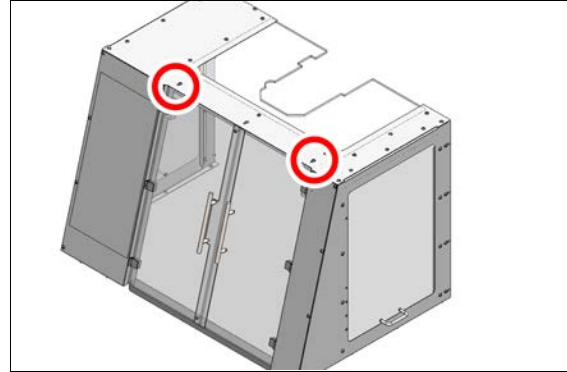


Figure 4-128: Cable tie holder locations for the top of the enclosure.

11. Use two M5 nuts to attach cable tie holders to the screws along the roof and on the inside of the enclosure.
12. Identify two M5 button head cap screws used to attach the enclosure to the chip pan as shown in the image.

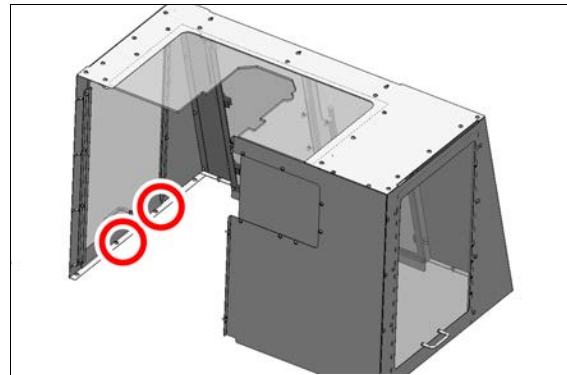


Figure 4-129: Cable tie holder locations for the bottom of the enclosure.

13. Temporarily remove the screw, and use it to attach two cable tie holders to the inside of the enclosure.
14. Route the wires from the door switches to the right side of the enclosure and toward the electrical cabinet.
15. Use the cable tie holders to secure all door switch wires in place.

2-Hinge Doors (Older Enclosure)

1. Power off the machine and the PathPilot controller.
 - a. Push in the machine's red Emergency Stop button, which removes power to motion control.
 - b. From the PathPilot interface, select **Exit**.
 - c. Turn the Main Disconnect switch to **OFF** on the side of the electrical cabinet.

2. Identify the 3-hole bracket that's included with this kit. You'll need to use it to replace the 2-hole bracket that's already installed onto the door switch.
3. Remove the two M3 x 0.5 - 6 button head cap screws from the door switch assembly with a 2 mm hex wrench.
4. Secure the reed switch onto the 3-hole mounting bracket with a 2 mm hex wrench.

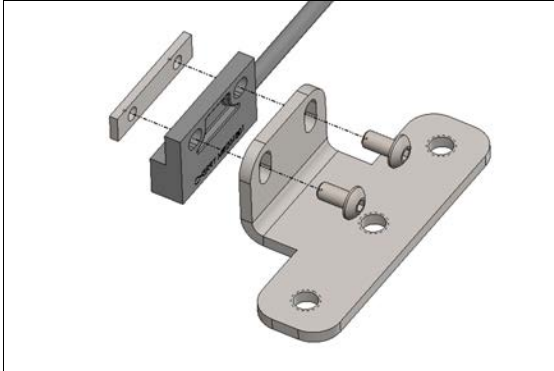


Figure 4-130: 3-hole door switch assembly.

5. Identify the orientation of the door switches:
 - The door switch with the **shorter** wires is installed on the right side of the enclosure.
 - The door switch with the **longer** wires is installed on the left side of the enclosure.
6. Open both doors on the enclosure.
7. Remove three 10-32 x 3/8-inch flat-head Phillips screws and three 10-32 nuts that secure the two top door hinges to the enclosure. Then, discard the screws, and set aside the nuts.
8. Align the holes on the door switch with the holes in the door hinge inside of the enclosure.
9. Use the three provided M5 screws to secure each door switch to the top door hinge.

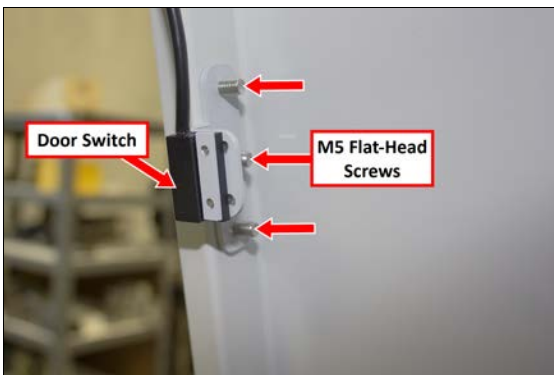


Figure 4-131: Example of the door switch installed as viewed from the inside of the enclosure.

10. Use a small, flat-blade screwdriver to loosen the set screws on each top door hinge.
11. Slide a magnet bracket between each door window and top door hinge.

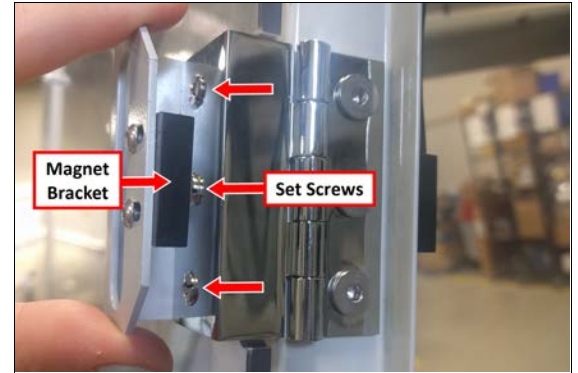


Figure 4-132: Example of an installed magnet bracket.

12. Tighten the screws on each door hinge to secure the magnet brackets to the door windows.

NOTICE! To prevent window damage from over-tightening, use your fingers to tighten the screws.

13. Close each door on the enclosure, and make sure that the magnet contacts the door switch.
14. Identify four screws on the enclosure as shown in the following image.

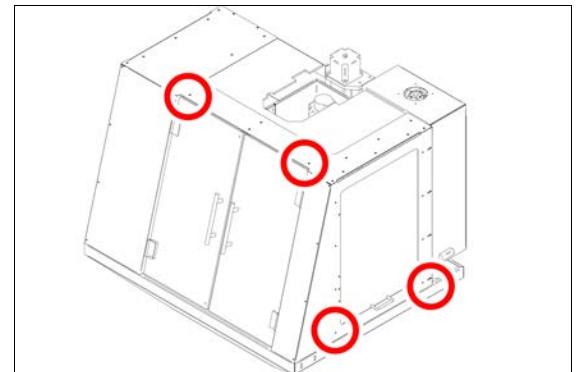


Figure 4-133: #10-32 Phillips head screw locations on the enclosure.

15. Use four #10-32 nuts that you set aside in Step 7 to attach four cable tie holders to the inside of the enclosure at the locations that you identified in the previous step.
16. Put four cable ties into the four cable tie holders.

4: INSTALLATION

4.7 Install Accessory Components

17. Route the wires from the door switches to the right side of the enclosure and toward the electrical cabinet.
18. Use the cable tie holders to secure all door switch wires in place.

Install the Safety Screws

1. Examine your machine:
 - If your enclosure has holes for safety screws pre-drilled above the side windows, go to Step 4.
 - If your enclosure doesn't have holes for safety screws pre-drilled, go to Step 2.
2. Mark the locations for the two safety screws on the left side of the enclosure as follows:
 - Each screw must be 1/2-inch below the top of the enclosure panel.
 - Allow approximately 9.4 inches between the safety screws.

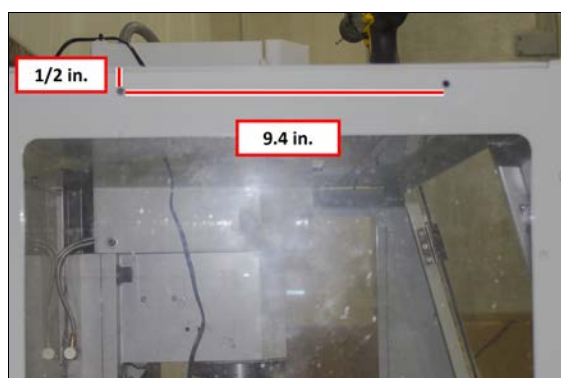


Figure 4-134: Example of the locations to drill holes for the safety screws.

3. Use a 1/4-inch drill bit to drill two holes in the locations that you marked in Step 2.
4. Insert one safety screw into each hole.
5. Use the Torx L-key and a 5/16-inch wrench to secure a lock nut to each safety screw.



Figure 4-135: Example of securing a lock nut to a safety screw.



Note: Keep the Torx L-key. You'll need it to remove the safety screws from the enclosure.

6. Repeat Step 1 through Step 5 to install two safety screws on the right side of the enclosure.

Make Electrical Connections

1. Use a 3 mm hex wrench to remove 10 socket head cap screws that secure the electrical cabinet panel to the back of the machine. Then, set all set aside.
2. Identify the existing access panel on the bottom of the electrical cabinet.

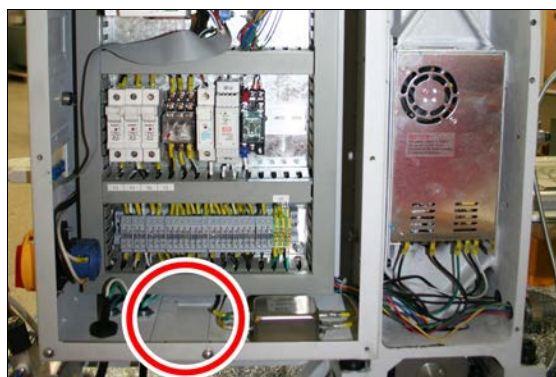


Figure 4-136: Access panel on the bottom of the electrical cabinet.

3. Use a 2 mm hex wrench to remove four socket head cap screws and the access panel. Then, set all aside.
4. Identify the access panel assembly provided.
5. Route the wires from the door switch through the push-connect fitting on the access panel assembly.

6. Use a 2 mm hex wrench to attach the new access panel assembly in place of the original access panel with four socket head cap screws that you set aside in Step 3.

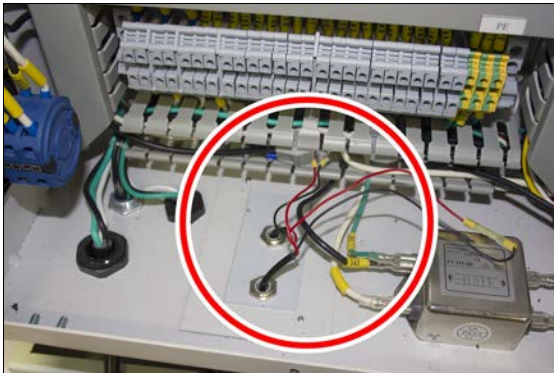


Figure 4-137: Example of a new access panel assembly installed on the bottom of the electrical cabinet.



Note: If you misplace these screws, there are four extra provided in this kit.

7. Remove the lower wire trough inside the electrical cabinet, and set it aside.
8. Locate the jumper wire between wire LN1 and G1 on the terminal block.



Figure 4-138: Example of the jumper wire on the terminal block.

9. Use a small, flat-blade screwdriver to loosen the screws above the LN1 and G1 slots on the terminal block, and then remove and discard the jumper wire.
10. Route wire LN1 from the door switch to slot LN1 on the terminal block. Then, insert it in place of the jumper wire, and tighten the screw above the slot.
11. Route wire G1 from the door switch to slot G1 on the terminal block. Then, insert it in place of the jumper wire, and tighten the screw above the slot.

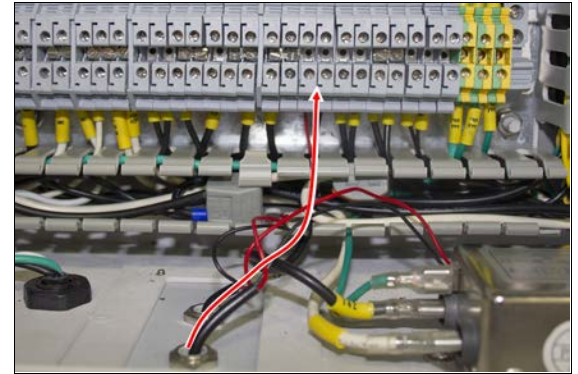


Figure 4-139: Example of the door switch wires installed on the terminal block.

12. Replace the lower wire trough that you set aside in Step 7.
13. Replace the electrical cabinet panel on the back of the machine that you set aside in Step 1.

Verify the Installation

1. Close both enclosure doors.
2. Power on the machine and the PathPilot controller.
 - a. Turn the PathPilot controller strip to **ON**.
 - b. Turn the Main Disconnect switch to **ON** on the side of the electrical cabinet.
 - c. Twist out the machine's red Emergency Stop button, which enables movement to the machine axes and the spindle.
 - d. Press the Start button.
3. From the PathPilot interface, on the Status tab, make sure that the **Door Switch** LED is off, which indicates that both enclosure doors are closed. If the **Door Switch** LED is on, go to Adjust Installed Components.



Figure 4-140: Door Switch LED on the Status tab.

4. Open both enclosure doors.
5. From the PathPilot interface, on the **Status** tab, make sure that the **Door Switch** LED is on, which indicates that both enclosure doors are open.

4: INSTALLATION

4.8 Set Up the PathPilot Controller

4.8 SET UP THE PATHPILOT CONTROLLER

Before operating your machine, configure in PathPilot the date, time, keyboard language.

4.8.1 Specify the Date and Time

1. From the PathPilot interface, on the **Main** tab, in the **MDI Line DRO** field, type `ADMIN DATE`. Then select the **Enter** key.

The **Time and Date Settings** dialog box displays.



Note: Before using any ADMIN commands, you must release the Emergency Stop. Rotate the Emergency Stop button one quarter turn to release it.

2. Complete the fields in the **Time and Date Settings** dialog box, and then select **Close**.

4.8.2 Specify the Keyboard Language

By default, the keyboard language is set to English.

To specify a different keyboard language:

1. From the PathPilot interface, on the **Main** tab, in the **MDI Line DRO** field, type `ADMIN KEYBOARD`. Then select the **Enter** key.
The **Keyboard Preferences** dialog box displays.
2. Select the **Layouts** tab and select the desired language.
If the language you want is not listed, select **Add** to specify the language. Then, select **Close**.

4.8.3 Configure the Optional Touch Screen Kit

Before using a touch screen, you must make sure that it's configured and calibrated. To calibrate it:

1. From the PathPilot interface, on the **Main** tab, in the **MDI Line DRO** field, type `ADMIN TOUCHSCREEN`. Then select the **Enter** key.
2. Follow the on-screen instructions.

4.8.4 Update PathPilot

We're constantly updating PathPilot to bring you more features. Before operating your machine, update to the latest version.

1. Confirm that the PathPilot controller is powered on and out of **Reset** mode.
2. Downloading and installing an update file requires an Internet connection. From the **Status** tab, confirm that

the **Internet** button LED light is on. (To configure the network, select the LED light.) Then, select **Update**.



Figure 4-141: Update button on the Status tab.

3. From the **Software Update** dialog box, select **Check Online**.

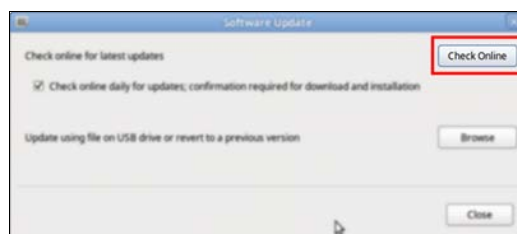


Figure 4-142: Software Update dialog box.

4. Select **Install**.



Figure 4-143: Install button on the Software Update dialog box.

The update file is downloaded, and a notification dialog box displays.

5. From the dialog box, select **OK**.
The update file is installed on the PathPilot controller.
6. Follow the on-screen instructions to restart the PathPilot controller.

SYSTEM BASICS

IN THIS SECTION, YOU'LL LEARN:

- About the main components of the machine and how it moves.



Before operating the machine in any way, you must read and understand this section.

CONTENTS

5.1 System Reference.....	84
5.2 Basic Controls Reference.....	85

5: SYSTEM BASICS

5.1 System Reference

5.1 SYSTEM REFERENCE

To operate your machine, you must become familiar with the components of its system.

5.1.1 Machine Table

The machine table is 18" × 6.3" (457 mm × 160 mm).

Don't put more than 150 lb (68 kg) on the machine table.

Always verify that the workpiece and workholding devices are centered on the machine table.

The machine table has three T-slots along the X-axis, used to secure workholding devices (on which to secure your workpiece while machining):

- **T-Slots** 3/8" (9.5 mm) (three slots)
- **T-Slot Spacing** 2-3/8" (60.3 mm)
- **Center T-Slot** Precision ground to 0.625" (15.9 mm)

5.1.2 Spindle

The machine spindle uses an R8 taper to hold Tormach Tooling System (TTS) tool holders.

- **Spindle Power** 0.75 hp (0.56 kW)

About the Spindle

The machine spindle gives power to the cutting tool, which allows it to remove material from the workpiece.

Operate the spindle either manually or by G-code commands (entered in the MDI Line DRO field or programmed into a G-code program). Manually controlling the spindle is useful when you're setting work offsets for tools that require the spindle to be running (like "wiggler"-style edgefinders or coaxial indicators).

The machine's spindle rotates clockwise (forward) at a specified spindle speed.

5.1.3 Axes

The machine has three linear axes of motion, and one optional rotary axis, used for machining:

- The X-axis, which is (horizontally) along the long side of the machine table.
- The Y-axis, which is (horizontally) along the short side of the machine table.
- The Z-axis, which is (vertically) along the mill column.
- *(Optional)* The A-axis, which is the axis of rotation that is typically aligned to the X-axis.

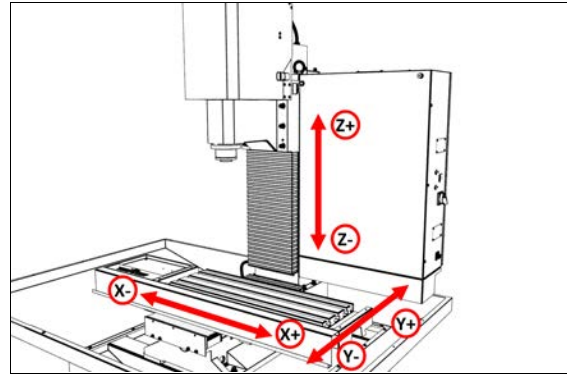


Figure 5-1: Axis directions on the machine.

Each axis has a different limit of travel, which is the distance it moves before reaching a limit switch:

- **X-Axis** 10" (254 mm)
- **Y-Axis** 6.25" (159 mm)
- **Z-Axis** 10" (254 mm)

5.2 BASIC CONTROLS REFERENCE

To safely and effectively operate your machine, you must become familiar with how it moves. The machine has two forms of basic controls: machine controls and the PathPilot interface.

5.2.1 Machine Controls

The following controls energize the machine's control electronics:

- The Main Disconnect switch, located on the right side of the electrical cabinet.

The Main Disconnect switch has two positions: **OFF** and **ON**. When it's in the **OFF** position, it separates the other machine control electronics from the mains electrical supply. When it's in the **ON** position, the other machine control electronics are able to receive power.



WARNING! Before opening the electrical cabinet for maintenance or troubleshooting, you must lockout the mains power: Turn the Main Disconnect switch to the **OFF** position, and secure an approved lockout device through the lockout rings at the bottom of the switch.

- The operator box — which contains the green Start button and the red Emergency Stop button — located on the right side of the chip tray.

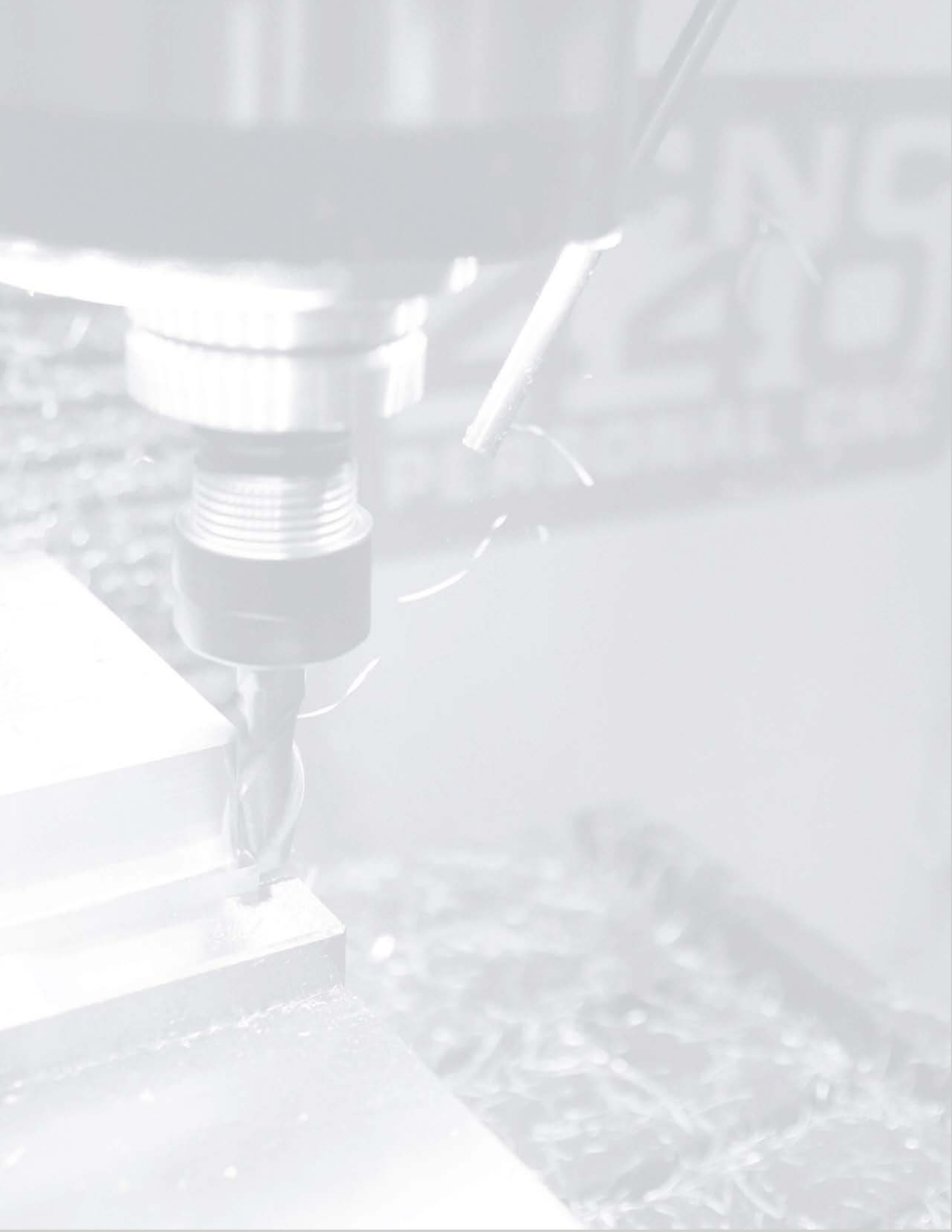
When pushed in, the Emergency Stop button interrupts power to the spindle and axis drives, and stops the machine's motion. When the Emergency Stop button is twisted out, press the Start button to enable the machine, allowing spindle and axis motion. The Start button's LED turns on when the machine is enabled and the spindle and axis drives receive power.

5.2.2 PathPilot Interface

PathPilot is the primary means by which you interact with your machine. PathPilot controls all of the automatic motion of the machine axes and spindle, as well as some accessories. The PathPilot control system consists of:

- Controller Arm
 - Controller
 - *(Optional)* Jog Shuttle
 - Keyboard

- Monitor or *(Optional)* Touch Screen Kit
- Mouse



PATHPILOT INTERFACE OVERVIEW

IN THIS SECTION, YOU'LL LEARN:

- How PathPilot is organized, and where you can access each tool or feature.

CONTENTS

6.1 About PathPilot	88
6.2 Notebook Section	89
6.3 Persistent Controls	92
6.4 Keyboard Shortcuts	94
6.5 Manage PathPilot Versions	95

6: PATHPILOT INTERFACE OVERVIEW

6.1 About PathPilot

6.1 ABOUT PATHPILOT

PathPilot is a combination hardware and software system that you use to control your machine. The controller hardware runs the PathPilot software.

The PathPilot interface is divided into sections: the Notebook section is in the top half of the screen, and the Persistent Controls section is in the bottom half.

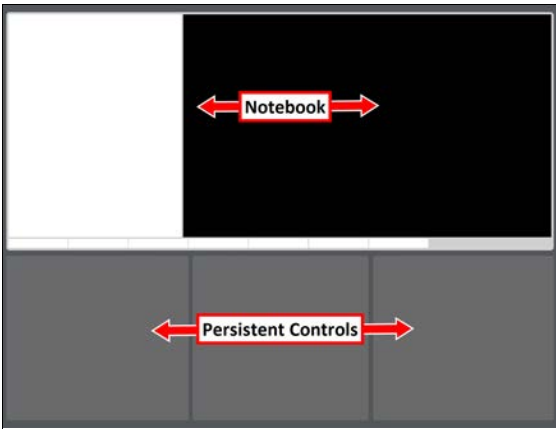


Figure 6-1: Sections in the PathPilot interface.

6.2 NOTEBOOK SECTION

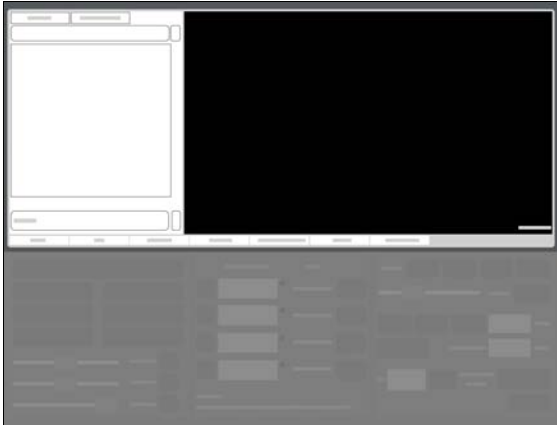


Figure 6-2: Notebook section.

The areas displayed in the Notebook section change depending on the activity that you're doing. Activities are grouped into the following tabs:

6.2.1 Main Tab	89
6.2.2 File Tab	89
6.2.3 Settings Tab	89
6.2.4 Offsets Tab	90
6.2.5 Conversational Tab	90
6.2.6 Probe Tab	90
6.2.7 Status Tab	90

6.2.1 Main Tab



Figure 6-3: Main tab.

By default, the Main tab is active when you power on the PathPilot controller. From the Main tab, you can do the following activities:

- Access G-code files that are already loaded into PathPilot, and open or close them.
For information, see "Access Recent G-Code Files" (page 99); "Close the Current Program" (page 99).
- Send G-code commands directly to the machine using the Manual Data Input (MDI) Line DRO field.
For information, see "Manually Enter Commands" (page 137).

- In a G-code program, do tasks like finding specific terms in the code, reading the code, or viewing the generated tool path.

For information, see "Search in the Code" (page 100); "Expand the G-Code Tab" (page 100); "Change the View of the Tool Path Display" (page 102).

6.2.2 File Tab



Figure 6-4: File tab.

From the File tab, you can do the following activities:

- Transfer G-code files into the PathPilot controller.
For information, see "Transfer Files to and From the Controller" (page 98).
- Edit G-code files.
For information, see "Edit G-Code" (page 99).
- Load .nc files into PathPilot to run a program.
For information, see "Load G-Code" (page 153).
- Move files within the system.
For information, see "Preview G-Code Files" (page 98); "Manage System Files" (page 143).

6.2.3 Settings Tab



Figure 6-5: Settings tab.

From the Settings tab, you can do the following activities:

- Change the network name with which you're using PathPilot.
For information, see "Change the Network Name" (page 116).

6: PATHPILOT INTERFACE OVERVIEW

6.2 Notebook Section

- Change the screen's layout orientation (landscape or portrait).
For information, see "Change the Screen Orientation" (page 116).
- Configure PathPilot for the accessories you're using.
For information, see "Enable the On-Screen Keyboard" (page 118); "Enable the USB M-Code I/O Interface Kit" (page 119); "Use a USB Camera" (page 119); "Specify the Tool Change Method" (page 117); .
- Turn on feeds and speeds suggestions when using conversational programming.
For information, see "Enable Feeds and Speeds Suggestions in Conversational Routines" (page 140).
- Specify the way in which you want to use a G30 move.
For information, see "Limit G30 Moves" (page 139).
- Identify the available G-code modes that you can use.
For information, see "View Available G-Code Modes" (page 128).

6.2.4 Offsets Tab



Figure 6-6: Offsets tab.

From the Offsets tab, you can do the following activities:

- Make and restore backup files of your settings.
For information, see "Create Backup Files" (page 143); "Restore Backup Files" (page 144).
- Import and export .csv files of your tool table.
For information, see "Import and Export the Tool Table" (page 145).
- Work with a table of tool descriptions and tool offsets.
For information, see "Set Tool Length Offsets" (page 155).
- Use an Electronic Tool Setter (ETS) to measure tools.
For information, see Use an to Measure Tools.
- Preset a G30 position.
For information, see "Use a G30 Position" (page 137).
- Read the currently programmed work offsets.
For information, see "View Work Offsets" (page 128).

6.2.5 Conversational Tab



Figure 6-7: Conversational tab.

From the Conversational tab, you can do the following activities:

- Use different functions to create simple G-code programs in PathPilot.
For information, see "About Conversational Programming" (page 103).

6.2.6 Probe Tab



Figure 6-8: Probe tab.

From the Probe tab, you can do the following activities:

- Configure and control a probe to help perform certain functions.
For information, see "Use a Probe with PathPilot" (page 123).

6.2.7 Status Tab



Figure 6-9: Status tab.

From the Status tab, you can do the following activities:

- View diagnostic machine information.
- Read error messages.

- Configure your internet connection.
For information, see "Enable an Internet Connection" (page 114).
- Update or install a previous version of PathPilot.
For information, see "Manage PathPilot Versions" (page 95).

6: PATHPILOT INTERFACE OVERVIEW

6.3 Persistent Controls

6.3 PERSISTENT CONTROLS



Figure 6-10: Persistent Controls section.

The areas that display in the Persistent Controls section don't change (unlike the Notebook section). They display regardless of the activity you're doing. Activities are grouped into the following areas:

6.3.1 Program Control Area.....	92
6.3.2 Position Status Area.....	92
6.3.3 Manual Control Area.....	92

6.3.1 Program Control Area

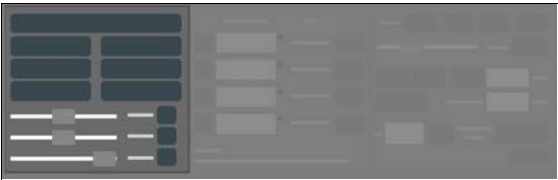


Figure 6-11: Program Control area.

From the Program Control area, you can do the following activities either before starting or while running a G-code program:

- Reset the machine.
For information, see "Bring the Machine Out of Reset" (page 129).
- Start, stop, or pause a G-code program.
For information, see "Start a Program" (page 131); "Stop Machine Motion" (page 131); "Use the Feed Hold Function" (page 133).
- Turn coolant on or off.
For information, see "Operate the Coolant Pump" (page 159).
- Use overrides to change the feed rate, spindle speed, and maximum velocity.

For information, see "Use the Feed Rate Override Function" (page 133); "Use the Maxvel Override Function" (page 134); "Use the Spindle Override Function" (page 135).

- Manually control a G-code program.
For information, see "Use M01 Break Mode" (page 134); "Use Single Block Mode" (page 135).

6.3.2 Position Status Area

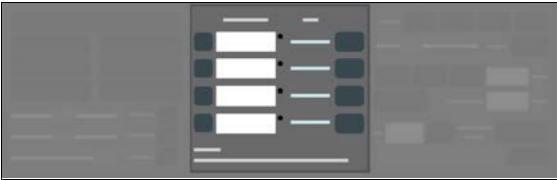


Figure 6-12: Position Status area.

From the Position Status area, you can do the following activities either before starting or after running a G-code program:

- Reference the machine axes.
For information, see "Reference the Machine" (page 149).
- Create work offsets.
For information, see "Set Work Offsets" (page 158).
- Understand how you're jogging the machine.
For information, see "View the Active Axis to Jog" (page 129); "View the Current Machine Position" (page 130); "View the Distance to Go" (page 132).
- Quickly determine which G-code modes are active.
For information, see "View the Active G-Code Modes" (page 132).

6.3.3 Manual Control Area



Figure 6-13: Manual Control area.

From the Manual Control area, you can do the following activities either before starting or after running a G-code program:

- Move the machine axes.
For information, see "Jog the Machine" (page 150).

- Change the spindle speed or feed rate.
For information, see "Change the Spindle Speed" (page 136); "Change the Feed Rate" (page 136).
- View or edit information about the current tool.
For information, see "Change the Tool Number" (page 136); "Use a G30 Position" (page 137) "View the Tool Length" (page 137).

6: PATHPILOT INTERFACE OVERVIEW

6.4 Keyboard Shortcuts

6.4 KEYBOARD SHORTCUTS

The following table lists the keyboard shortcuts in PathPilot.

Keyboard Shortcut	Use to...
Alt+E	Edit the currently loaded G-code program (from any tab in the PathPilot interface)
Alt+Enter	Use the Manual Data Input (MDI) Line DRO field
Alt+F	Use the coolant function
Alt+R	Start a program
Esc	Stop a program
Shift+Alt+E	From the Main tab, quickly edit a G-code program with conversational programming
Space Bar	Feed hold the machine

6.5 MANAGE PATHPILOT VERSIONS

You don't need to install updates sequentially. You can update from any previous version to the current version of PathPilot. Depending on what you want to do, refer to the following sections:

- "Download and Install an Update File from the Controller" (below)
- "Install an Update File from a USB Drive" (below)
- "Install a Previous Version of an Update File" (on the next page)

6.5.1 Download and Install an Update File from the Controller

1. Confirm that the PathPilot controller is powered on and out of **Reset** mode.
2. Downloading and installing an update file requires an Internet connection. From the **Status** tab, confirm that the **Internet** button LED light is on. (To configure the network, select the LED light.) Then, select **Update**.



Figure 6-14: Update button on the Status tab.

3. From the **Software Update** dialog box, select **Check Online**.

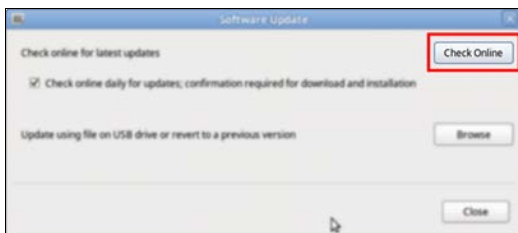


Figure 6-15: Software Update dialog box.

4. Select **Install**.



Figure 6-16: Install button on the Software Update dialog box.

The update file is downloaded, and a notification dialog box displays.

5. From the dialog box, select **OK**.
The update file is installed on the PathPilot controller.
6. Follow the on-screen instructions to restart the PathPilot controller.

6.5.2 Install an Update File from a USB Drive

1. From the PathPilot support center, download the most recent PathPilot update file.
2. Transfer the PathPilot update file to a USB drive.
3. Put the USB drive into the PathPilot controller.
4. Confirm that the PathPilot controller is powered on and out of **Reset** mode.
5. From the **Status** tab, select **Update**.



Figure 6-17: Update button on the Status tab.

6. From the **Software Update** dialog box, select **Browse**.

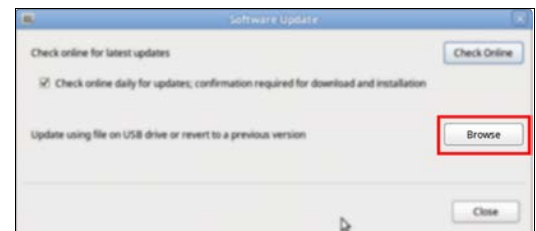


Figure 6-18: Software Update dialog box.

6: PATHPILOT INTERFACE OVERVIEW

6.5 Manage PathPilot Versions

7. From the **Browse** dialog box, select **USB**.



Figure 6-19: Browse dialog box.

8. Select the desired update file, and then select **Update**.
The update file is installed on the PathPilot controller.
9. Follow the on-screen instructions to restart the PathPilot controller.

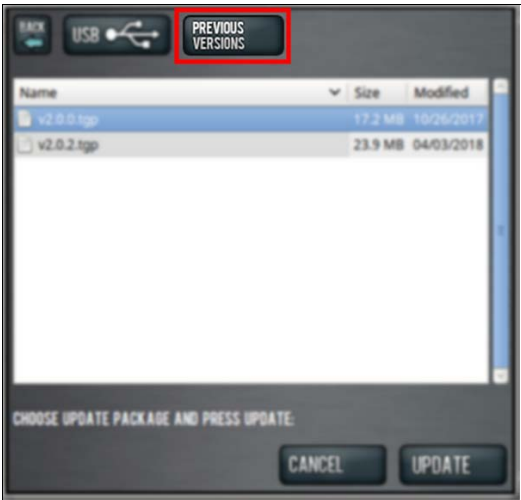


Figure 6-22: Browse dialog box.

5. Select the desired update file, and then select **Update**.
The update file is installed on the PathPilot controller.
6. Follow the on-screen instructions to restart the PathPilot controller.

6.5.3 Install a Previous Version of an Update File

1. Confirm that the PathPilot controller is powered on and out of **Reset** mode.
2. From the **Status** tab, select **Update**.



Figure 6-20: Update button on the Status tab.

3. From the **Software Update** dialog box, select **Browse**.

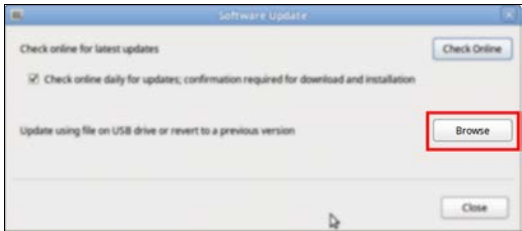


Figure 6-21: Software Update dialog box.

4. From the **Browse** dialog box, select **Previous Versions**.

PATHPILOT TOOLS AND FEATURES

IN THIS SECTION, YOU'LL LEARN:

- How to use PathPilot, depending on the activity that you want to do.

CONTENTS

7.1 Create and Load G-Code Files	98
7.2 Machine Settings and Accessories	114
7.3 Set Up G-Code Programs	123
7.4 Run G-Code Programs	129
7.5 Control G-Code Programs	133
7.6 System File Management	143

7: PATHPILOT TOOLS AND FEATURES
7.1 Create and Load G-Code Files

7.1 CREATE AND LOAD G-CODE FILES

To get started with PathPilot, you must first load or create a G-code file.

7.1.1 Load G-Code 98
7.1.2 Edit G-Code 99
7.1.3 Read G-Code 100
7.1.4 Use Conversational Programming 103

7.1.1 Load G-Code

To run a G-code program on a PathPilot controller, you must first verify that the file is on the controller. For more information on transferring and moving files, see "Transfer Files to and From the Controller" (below).

To load G-code:

- 1. From the File tab, in the Controller Files window, select the desired .nc file.
- 2. Select Load G-Code.

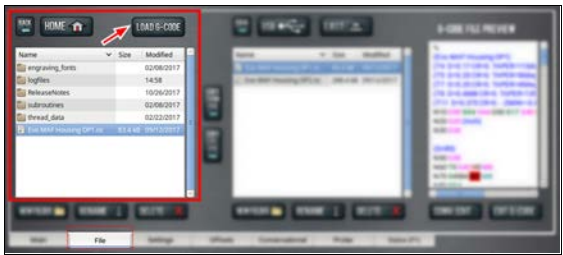


Figure 7-1: Controller Files window on the File tab.



Note: This function is only available for files stored on the PathPilot controller.

PathPilot loads the G-code file and opens the Main tab.

Transfer Files to and From the Controller

To run a G-code program, you must transfer the files to the PathPilot controller. For information, see "About System Files" (page 143).

To transfer files to and from the controller:

- 1. Insert a USB drive into any open USB port.
- 2. From the File tab, select the file to transfer (either in the USB Files window or the Controller Files window).



Note: Select Back to move backward and either Home or USB to move to the highest level.

- 3. Select the location to which you want to copy the transferred file.

- 4. Select either Copy From USB or Copy to USB.



Figure 7-2: File tab.



Note: The file must have a unique name. If it doesn't, you must either overwrite the file, rename the file, or cancel the file transfer.

- 5. Select Eject.
It's safe to remove the USB drive from the controller.

About System Files

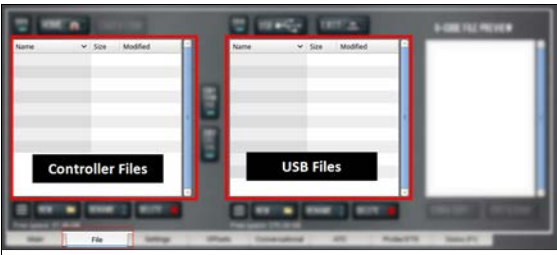


Figure 7-3: File tab.

PathPilot doesn't run G-code program files from a USB drive. You must first transfer files to the PathPilot controller. For information on transferring files, see "Transfer Files to and From the Controller" (above).

Preview G-Code Files

You can preview an .nc file that's either on the PathPilot controller or on a USB drive.

To preview G-code files:

- From the File tab, in the Controller Files window or the USB Files window, select an .nc file.
The text displays in the Preview window.



Figure 7-4: File tab.

Access Recent G-Code Files

You can load a recently loaded G-code file from the Main tab. For information, see "About the G-Code Tab" (on the next page).

To access recent G-code files:

1. From the **Main** tab, in the **G-Code** tab, select the **Recent Files** menu.

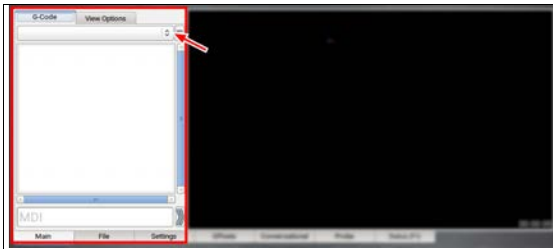


Figure 7-5: Recent Files menu on the Main tab.

The last five program files loaded into PathPilot display.

2. Select the name of the desired G-code program.
The G-code program loads.

Close the Current Program

1. From the **Main** tab, on the **G-Code** tab, select the **Recent Files** menu.
2. Select **Clear Current Program**.



Figure 7-6: Recent Files menu on the Main tab.

The currently loaded G-code program closes.

7.1.2 Edit G-Code

In PathPilot, there are two ways to edit G-code:

- Edit G-Code with a Text Editor** 99
- Edit G-Code with Conversational Programming** 99

Edit G-Code with a Text Editor

You can edit .nc files that are on the PathPilot controller. If the .nc file is in the USB Files window, you must first transfer it to the controller; go to "Transfer Files to and From the Controller" (on the previous page).

To edit G-code with a text editor:

1. From the **Controller Files** window, highlight the .nc file and select **Edit G-code**.



Figure 7-7: Edit G-code button on the File tab.

The file opens in a text editor.

2. Make and save the appropriate changes to the file.
3. Close the text editor.

Tip! To quickly edit an already loaded G-code program from the Main tab, you can use a keyboard shortcut: Shift+Alt+E.

Edit G-Code with Conversational Programming

You can edit .nc files that are on the PathPilot controller. If the .nc file is in the USB Files window, you must first transfer it to the controller; go to "Transfer Files to and From the Controller" (on the previous page).

To edit G-code with conversational programming:

1. From the **File** tab, select the .nc file.
2. Select **Conv. Edit**.

The file opens in a job assignment editor window: the program's job assignments are on the left and a preview of the program is on the right.

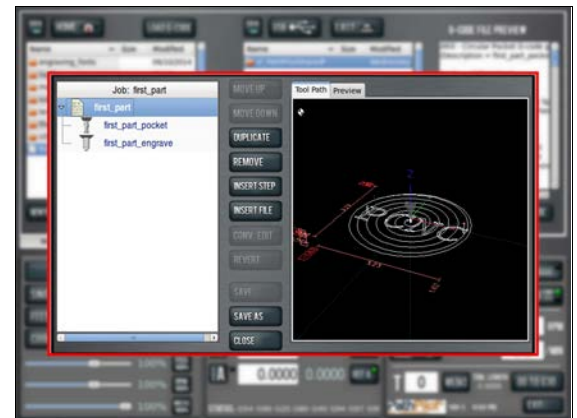


Figure 7-8: Job assignment editor.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

3. Edit the file contents as needed. Do any of the following:

Change the Step Order.....	100
Create a New Job Assignment	100
Load an Existing G-Code File.....	100
Edit a Job Assignment	100

4. Select **Save**.
The G-code program file is updated.

Change the Step Order

- Select **Move Up, Move Down, Duplicate, or Remove**.

Create a New Job Assignment

1. Select **Insert Step**.
PathPilot opens the **Conversational** tab.
2. Create the new job assignment.
3. Select **Insert**.
4. (Optional) Edit the job assignment order in the program.

Load an Existing G-Code File

1. Select **Insert File**. You can insert G-code files that are hand-written, generated from CAM software, or generated from conversational programming in PathPilot.
2. Navigate to and select the .nc file that you want to insert.
3. Select **Open**.
4. (Optional) Edit the job assignment order in the program.

Edit a Job Assignment

1. Select the job assignment, and then select **Conv. Edit**.
PathPilot opens the **Conversational** tab.
2. Edit the job assignment.
3. Select **Finish Editing**.

Tips

- To restore an edited job assignment to its original parameters: select **Revert**.



Note: **Revert** is only available for individual job assignments created in conversational programming.

- To undo all changes made to an entire G-code program: select **Close**. When prompted, select **Close Without Saving**.

7.1.3 Read G-Code

Once your G-code file is loaded into PathPilot, you can read it in the following ways:

Expand the G-Code Tab	100
Search in the Code	100
Set a New Start Line	101
Change the View of the Tool Path Display.....	102

Expand the G-Code Tab

You can change the size of the G-Code tab if you need more space to view the code. For more information on using the G-Code tab, see "About the G-Code Tab" (below).

To expand the G-Code tab:

- Select the **Window Expander**.

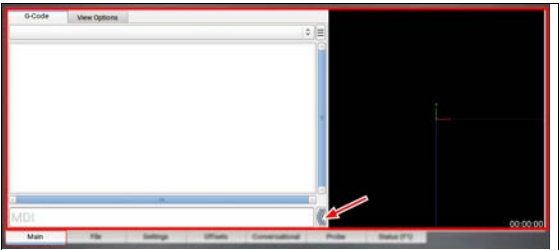


Figure 7-9: Window Expander on the Main tab.

The **Tool Path** display shrinks.

About the G-Code Tab

The G-Code tab displays the code of the currently loaded program file. Use the scroll bars to view the entire file. You can make the G-Code tab larger. For information, see "Expand the G-Code Tab" (above).

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

- **Green Line** Indicates the start line (the line from which PathPilot starts the program).
To change the start line, go to "Set a New Start Line" (on the next page).
- **Orange Line** Indicates the line of code that PathPilot is currently executing.

Search in the Code

You can use PathPilot to search the text of a G-code program file for specific numbers, codes, or other items of interest (like tools, feeds, and speeds).

To search in the code:

1. From **Main** tab, on the **G-Code** tab, select any line of code to use as a starting point.
2. In the **MDI Line DRO** field, type **FIND** followed by one of the following:
 - Any text. PathPilot searches for instances of the specific number or code.

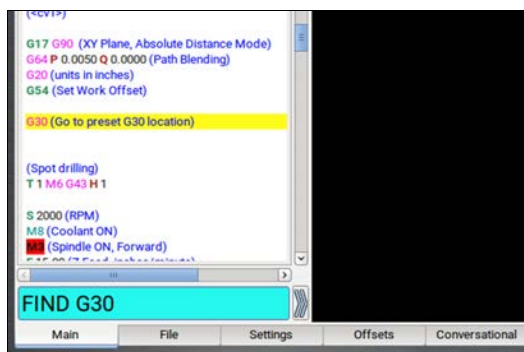


Figure 7-10: Search for a text command.

- **FEED**. PathPilot searches for instances of the actual word **Feed** and any **F** G-code command.

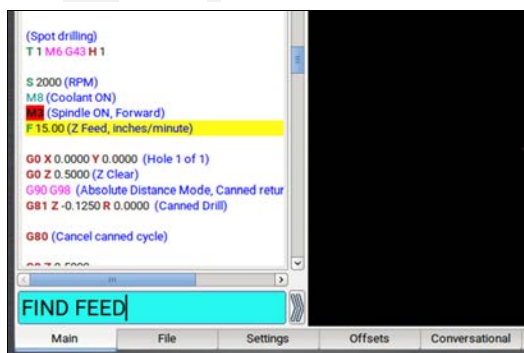


Figure 7-11: Search for a feed command.

- **SPEED**. PathPilot searches for instances of the actual word **Speed** and any **S** G-code command.
- **TOOL**. PathPilot searches for instances of the word **Tool** and any **T** G-code command.



Note: The find command is not case-sensitive.

3. Select the **Enter** key.
If PathPilot finds the information, the searched term is scrolled to and highlighted in the **G-Code** tab.
4. (Optional) Select **Enter**.
PathPilot finds the next instance of the searched text.
5. (Optional) Select **Enter+Shift**.

PathPilot finds the previous instance of the searched text.



Note: When the search reaches the end of the G-code file, it starts again from the beginning.

Set a New Start Line

The start line (the line from which PathPilot starts the program) is, by default, the first line of code in the program. To set a new start line:

1. From the **Main** tab, on the **G-Code** tab, do one of the following:
 - Right-click any line in the program.

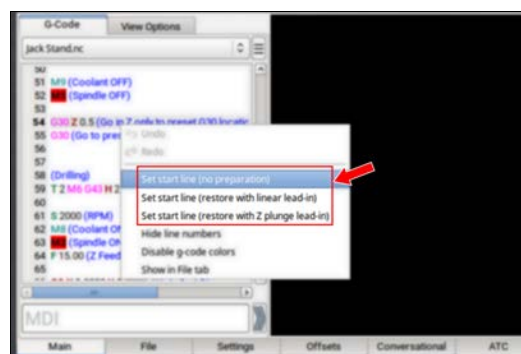


Figure 7-12: Accessing the Options menu by right-clicking.

- Tap the line. Then, select the **Options** menu.
2. Select the desired lead-in move. For information, see "Lead-In Moves" (below).

Lead-In Moves

- **Set start line (no preparation)** Keep the current tool in the spindle, with the current tool length applied. The machine executes the start line from the current position.



Note: We don't recommend this option for starting partway through a cut.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

EXAMPLE

- Starting the program at a tool change.
- Starting the program with a different tool in the spindle than the program calls for (like if your tool broke, which you've replaced, but you'd rather not edit the entire program or the tool table entry).

- Set start line (restore with linear lead-in)** Perform a tool change (as required). The machine rapids in X and Y, then Z to the current position, then feeds in a straight linear line to the start line position.

Note: This option assumes that the current position is the lead-in position.

EXAMPLE

Quickly resuming work after stopping the program to make an adjustment to the machine setup (like clearing chips, removing an object, or turning on the coolant pump). Because the machine's already set up, you can position the tool near the stopping point.

- Set start line (restore with Z plunge lead-in)** Perform a tool change (as required). The machine rapids in Z to G30 clearance height, rapids in X and Y to the start line position, then feeds in Z to the start line position.

EXAMPLE

Running a sub-section of a large program when the correct tool isn't loaded (and positioning the tool tip near the starting point is difficult, like with a long tool or fly cutter loaded). This option doesn't require you to jog to the exact lead-in position.

Change the View of the Tool Path Display

- From the **Main** tab, do one of the following:
 - Right-click the **Tool Path** display.

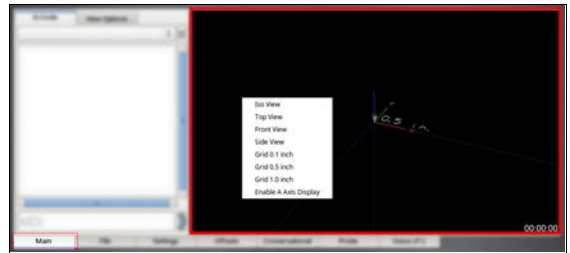


Figure 7-13: Tool Path display on the Main tab.

- Select the **View Options** tab.

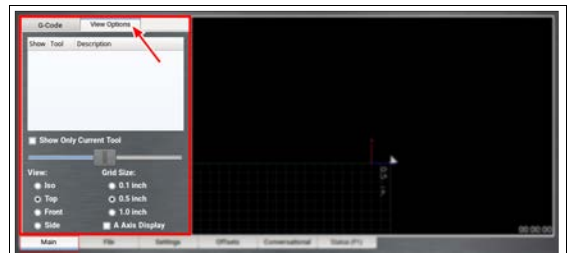


Figure 7-14: View Options tab on the Main tab.

- Select a new view.
For information, see "About the Tool Path Display" (below).

About the Tool Path Display

The Tool Path display is a graphical representation of the currently loaded G-code file's tool path.

There are four available views:

- Front
- Iso
- Side
- Top

You can see grid lines behind the tool path while you are using either a Top, Front, or Side view. Depending on which programming mode you're in (G20 or G21), PathPilot defaults to one of the following grid line spacings:

- G20 Mode** 1/2 in. intervals
- G21 Mode** 5 mm intervals

In the Tool Path display, there are four different line types:

- Dotted Blue Lines** Indicate the boundary box (the ends of travel of the axes).

- **Red Lines** Indicate the tool path as it is cut.



Note: The Tool Path display shows the program extents — the furthest points to which the tool will travel while running the program — of the currently loaded G-code file alongside the tool path lines.

- **White Lines** Indicate the preview lines.
- **Yellow Lines** Indicate the jogging moves.

To erase the jogging moves (yellow line) or the tool path (red lines), do one of the following:

- Double-click anywhere in the Tool Path display.
- Select Reset.

7.1.4 Use Conversational Programming

To create simple parts, use the conversational programming feature in PathPilot.

About Conversational Programming.....	103
Create a Face on a Part.....	103
Create a Profile on a Part.....	104
Create a Pocket on a Part.....	106
Create Hole Locations on a Part (Drill).....	108
Create Threads on a Part.....	110
Create Text to Engrave on a Part.....	111
Import a DXF File.....	112

About Conversational Programming

PathPilot includes G-code generators intended to make simple G-code programs:

- Programs for simple parts.
- Programs for parts made up of a collection of simple features.



Note: For complex parts, or parts with complex shapes, we recommend you use a CAD/CAM program.

The Conversational tab is divided into two sections:

- Parameters common to most operations, like speeds and feeds.



Note: DRO fields that are grayed out are not available for the specific conversational features.

- Parameters specific to each operation, like part geometry.

Create a Face on a Part

Using conversational programming, you can program PathPilot to take multiple cuts — each following the last — on an X/Y plane over a Z range. For information, see "About Facing" (on the next page).

Before You Begin

Before you begin, you must verify that you enter the program values considering the following:

- The area from the rear, left corner of the workpiece to the rear, right corner of the workpiece must be clear of any obstructions from the **Z Start** position to the **Z End** position.
- The top of the workpiece must be free of any workholding devices.
- The value used in the **Z End** DRO field must be such that it is above the workholding device.
- The value used in the **Z Clear** DRO field must be such that it is above any obstruction in the tool path between the end of one pass and the beginning of the next.

To create a face on a part:

1. From the **Conversational** tab, select the **Face** tab.
2. From the **Conversational DROs** group, set the parameters for the facing operation.



Figure 7-15: Conversational DROs on the Face tab.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

3. Work through the program-specific DRO fields:
 - a. In the **X Start** DRO field and the **X End** DRO field, type the location of the workpiece edges. We recommend using the rear left corner of the part for the **X Start** value.
 - b. In the **Y Start** DRO field and the **Y End** DRO field, type the location of the workpiece edges. We recommend using the front right corner of the part for the **Y End** value.
 - c. In the **Stepover** DRO field, type the required distance between tool paths. To prevent uncut areas in the spiral corners, we recommend limiting this value to 80% of the tool diameter. For information, see "Facing Reference" (below).
 - d. In the **Z Start** DRO field and the **Z End** DRO field, type the location of the first and last Z passes. For a single Z pass at the location typed in the **Z End** DRO field, type a value of 0 or a full Z range value into the **Depth of Cut** DRO field.
 - e. In the **Depth of Cut** DRO field, type the desired amount of material to remove.



Note: The depth of cut is later adjusted within the Z range so that each pass in the Z range has the same depth (rather than the last Z pass having a short depth of cut).

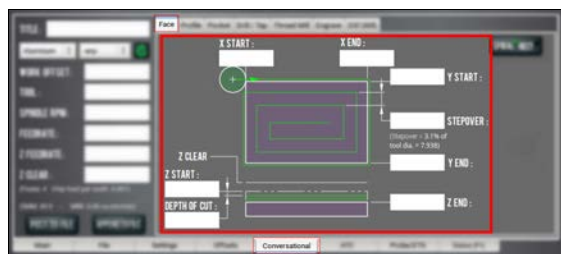


Figure 7-16: Program-specific DROs on the Face tab.

About Facing

Face milling is the process of cutting a surface that's perpendicular to the axis of the cutting tool. A facing program is usually used to cut an accurate, finished top surface on a rough piece of stock material. After a facing program is complete, tool marks remain — creating a fairly flat surface, with microscopic height differences.

Facing in PathPilot

When using a facing routine, each tool pass along the X-/Y-axis begins off to the side of the workpiece to avoid plunging into the workpiece. To compensate for this procedure, PathPilot sets lead-in tool paths outside of the workpiece using the part's work offsets, the tool's diameter, and the predetermined stepover value. PathPilot also adjusts the depth of cut to make sure each tool pass has the same depth, rather than cutting a short depth on the last pass of the program.

During a facing routine, PathPilot does the following:

1. Moves the machine to the predefined G30 position, or the tool change position.
2. If required, performs or requests a tool change.
3. Makes a rapid move in the X and Y direction to the beginning of the workpiece.
4. Makes a rapid move in the Z direction to the predefined Z Clear position.
5. Begins the cut in the X/Y plane at an adjusted Z depth of cut.



Note: The value entered into the Depth of Cut DRO field is adjusted within the Z range (the value entered into the Z End DRO field minus the value entered into the Z Start DRO field).

6. Makes cuts in a rectangular spiral from the workpiece perimeter to the workpiece center.

For information on using conversational programming in PathPilot to face a part, see "Create a Face on a Part" (on the previous page).

Facing Reference

PathPilot uses the following terms when creating a face on a part in conversational programming:

- **Stepover** Indicates how much space PathPilot creates between each spiral tool path.
- **Z Clear** Indicates the Z location that the tool moves (retracts) to when starting or ending a tool pass.

Create a Profile on a Part

Using conversational programming, you can program PathPilot to take multiple cuts — each following the last — on an X/Y plane over a Z range to form a boss. For information, see "About Profiling" (on the next page).

Before You Begin

Before you begin, you must verify that you enter the program values considering the following:

- The value used in the **Radius** DRO field must be between 0 and either:
 - One half of the boss' narrow width, or
 - The full radii on the long ends of the boss
- The value used in the **Z Clear** DRO field must be set to clear any obstructions between path changes.

To create a profile on a part:

1. From the **Conversational** tab, select the **Profile** tab.
2. From the **Conversational DROs** group, set the parameters for the profiling operation.



Figure 7-17: Conversational DROs on the Profile tab.

3. Work through the program-specific DRO fields:
 - a. In the **X Start** DRO field and the **X End** DRO field, type the location of the workpiece edges.
 - b. In the **Y Start** DRO field and the **Y End** DRO field, type the location of the workpiece edges.
 - c. In the **X Profile Start** DRO field, the **X Profile End** DRO field, the **Y Profile Start** DRO field, and the **Y Profile End** DRO field, type the location of the profile's outer edges.
 - d. In the **Stepover** DRO field, type the required distance between tool paths. If you want a single cut in the workpiece (to create a slot), type 0.
 - e. In the **Radius** DRO field, type the required radius for the corners of the profile. For no corner radius, type 0.
 - f. In the **Z Start** DRO field and the **Z End** DRO field, type the location for the first and last Z passes. For a single Z pass at Z End, type 0 — or a full Z range value — in the **Z DOC** DRO field.
 - g. In the **Z DOC** DRO field, type the desired amount of material to remove.

Note: The depth of cut is later adjusted within the Z range so that each pass in the Z range has the same depth (rather than the last Z pass having a short depth of cut).

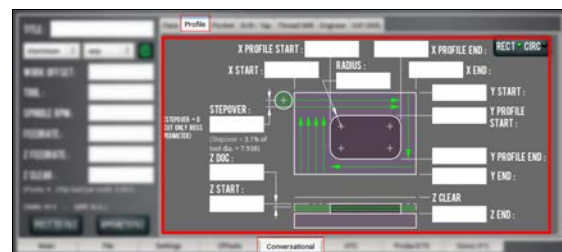


Figure 7-18: Program-specific DROs on the Profile tab.

About Profiling

A profiling program is usually used to create a circular or rectangular boss within a larger piece of stock material. The outer bound of the area is the stock material's perimeter. The inner bound of the area is the boss' perimeter.

Profiling in PathPilot

When using a profile routine, each tool pass along the X-/Y-axis begins off to the side of the workpiece to avoid plunging into the workpiece. To compensate for this procedure, PathPilot sets lead-in tool paths outside of the workpiece using the part's work offsets, the tool's diameter, and the predetermined stepover value. PathPilot also adjusts the depth of cut to make sure each tool pass has the same depth, rather than cutting a short depth on the last pass of the program. When you're creating a profile on a part using conversational programming, PathPilot does the following in the order listed:

1. Retracts the tool to the **Z Clear** position.

Note: The first Z pass cuts at **Z Start** minus Depth of Cut adjusted.

2. Makes a rapid movement to the beginning of the section.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

3. If required, makes a tool path around the perimeter of the boss to cut the programmed radii.



Note: Radius cuts use an adjusted feed rate to compensate for the difference between the tool's control point rate (at the tool center) and the actual rate at the radius surface.

4. Repeats Steps 1-3 for each predefined **Z Depth of Cut**.



Note: The last Z pass will cut at the **Z End** location.

For information on using conversational programming in PathPilot to create a profile, see "Create a Profile on a Part" (page 104).

Profiling Reference

PathPilot uses the following terms when creating a profile on a part in conversational programming:

- **Stepover** Indicates the tool path offset between section sweeps.
- **Z Clear** Indicates the Z location the tool moves to or retracts to when starting or ending a section change, section sweep, or Z pass.

Create a Pocket on a Part

Using conversational programming, you can program PathPilot to take multiple cuts — each following the last — on an X/Y plane over a Z range to form a circular or rectangular pocket. For information, see "About Pockets" (on the next page).

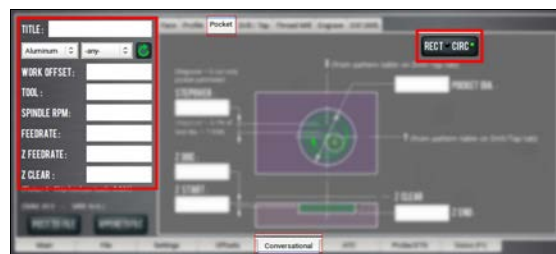
Before You Begin

Before you begin, you must verify that you enter the program values considering the following:

- The value used in the **Radius** DRO field must be between 0 and either:
 - One half of the pocket's narrow width, or
 - The full radii on the long ends of the pocket

To create a circular pocket on a part:

1. From the **Conversational** tab, select the **Pocket** tab.
2. From the **Conversational DROs** group, set the parameters for the pocket operation.



3. Work through the program-specific DRO fields:
 - a. In the **X Center** DRO field and the **Y Center** DRO field, type the location of the pocket's center.
 - b. In the **Pocket Dia.** DRO field, type the required diameter for the pocket.

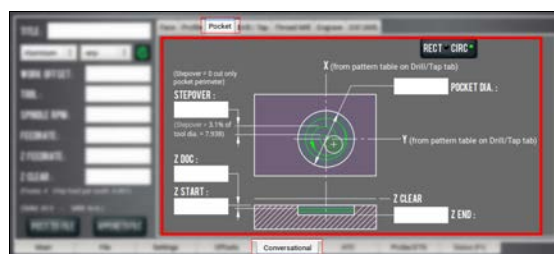


Note: The radius of the tool is used to set the tool path diameter.

- c. In the **Stepover** DRO field, type the required distance between each rotation of the spiral cut. For a single cut around the inside perimeter of the pocket (to create a slot), type 0.
- d. In the **Z Start** DRO field and the **Z End** DRO field, type the location of the first and last Z passes. For a single Z pass at the location typed in the **Z End** DRO field, type a value of 0 or a full Z range value into the **Depth of Cut** DRO field.
- e. In the **Z DOC** DRO field, type the desired amount of material to remove.



Note: The depth of cut is later adjusted within the Z range so that each pass in the Z range has the same depth (rather than the last Z pass having a short depth of cut).



To create a rectangular pocket on a part:

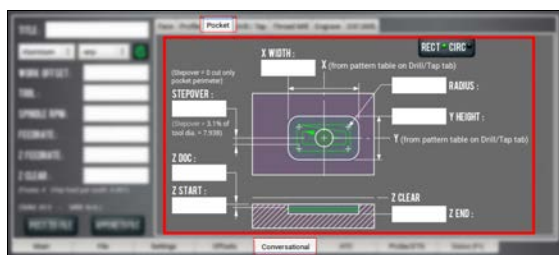
1. From the **Conversational** tab, select the **Pocket** tab.
2. From the **Conversational DROs** group, set the parameters for the pocket operation.



3. Work through the program-specific DRO fields:
 - a. In the **X Start** DRO field, the **X End** DRO field, the **Y Start** DRO field, and the **Y End** DRO field, type the location of the pocket edges.
 - b. In the **Radius** DRO field, type the required radius for the corners of the pocket. For no corner radius, type 0.
 - c. In the **Stepover** DRO field, type the required distance between adjacent tool paths. For a single cut around the inside perimeter of the pocket (to create a slot), type 0.
 - d. In the **Z Start** DRO field and the **Z End** DRO field, type the location of the first and last Z passes. For a single Z pass at the location typed in the **Z End** DRO field, type a value of 0 or a full Z range value into the **Depth of Cut** DRO field.
 - e. In the **Z DOC** DRO field, type the desired amount of material to remove.



Note: The depth of cut is later adjusted within the Z range so that each pass in the Z range has the same depth (rather than the last Z pass having a short depth of cut).



About Pockets

A pocket program is usually used to remove a large amount of material from a part.

Creating Pockets in PathPilot

For more information about creating a pocket on a part using conversational programming, see of the following sections, depending on the shape of the pocket:

- "About Circular Pockets" (below)
- "About Rectangular Pockets" (below)

For information on using conversational programming in PathPilot to make a pocket on a part, see "Create a Pocket on a Part" (on the previous page).

About Circular Pockets

PathPilot does one of the following, depending on the diameter of the tool:

- **Tool Diameter Larger Than Pocket Diameter**
PathPilot displays an error and does not create any G-code. You must select a different tool or edit the diameter of the pocket.
- **Tool Diameter Just Small Enough to Fit Within Pocket Diameter** PathPilot does the following in the order listed:
 1. Uses a straight Z plunge into the pocket center.
You must use a center-cutting end mill.
 2. Makes a single pass around the perimeter of the pocket at the adjusted Z depth of cut.
 3. Continues to make single passes around the perimeter of the pocket.
- **Pocket Diameter More Than Two Times Tool Diameter** PathPilot does the following in the order listed:
 1. Uses a helical entry into the pocket center.
 2. Makes a hole that is two times the diameter of the tool diameter in the pocket center.
 3. Makes a spiral cut out from the pocket center to the pocket diameter.
 4. Makes a cut around the perimeter of the pocket.

About Rectangular Pockets

PathPilot does one of the following, depending on the diameter of the tool:

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

- **Tool Diameter Larger Than Pocket Width** PathPilot displays an error and does not create any G-code. You must select a different tool or edit the width of the pocket.
- **Tool Diameter Just Small Enough to Fit Within Pocket Parameters** PathPilot does the following in the order listed:
 1. Uses a straight Z plunge into the pocket center. You must use a center-cutting end mill.
 2. Makes a single pass around the perimeter of the pocket at the adjusted Z depth of cut.
 3. Continues to make single passes around the perimeter of the pocket.
- **Length of Pocket More Than Two Times Diameter of Tool** PathPilot uses a linear ramp entry.
- **Pocket Parameters More Than Two Times Diameter of Tool** PathPilot uses a helical entry.

Create Hole Locations on a Part (Drill)

Using conversational programming, you can program PathPilot to make multiple holes on a part. For information, see "About Drilling" (on the next page).

Before You Begin

Before you begin, you must verify that you enter the program values considering the following:

- In the **Conversational DROs** group, the value in the **Z Clear** DRO field must be set to clear any obstructions between hole changes.

To make a specific hole pattern of evenly spaced holes around a circumference (also known as a bolt pattern):

1. From the **Conversational** tab, select the **Drill** tab.

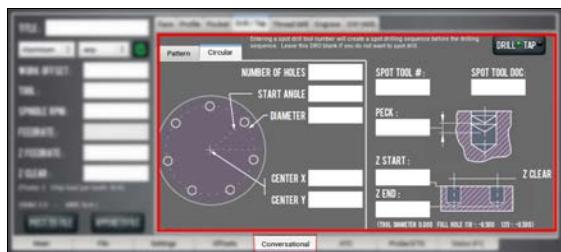


Figure 7-19: Drill tab on the Conversational tab.

2. From the **Drill** tab, select the **Circular** tab.
3. In the **Number of Holes** DRO field, type the number of holes (greater than 0) required for the pattern.

4. In the **Start Angle** DRO field, type the value of the angle from angle 0 (from **-90** to **90**).
 5. In the **Diameter** DRO field, type the size of the circular pattern (as defined by a line through the center point of each hole).
 6. In the **Center X** DRO field and the **Center Y** DRO field, type the location for the center of the hole.
 7. Go to "Create a Drilling Sequence" (on the next page).
- Use the Location table to make a list of X and Y locations for each hole using the same tool, the same Z location, and the same values in the Conversational DROs group.

Note: You can create holes using different tools or different parameters. You must first post the first group of hole locations, enter the second group of hole locations, and then append the second group to the existing posted file.

To make a hole pattern based on X and Y locations on a part:

1. From the **Conversational** tab, select the **Drill** tab.

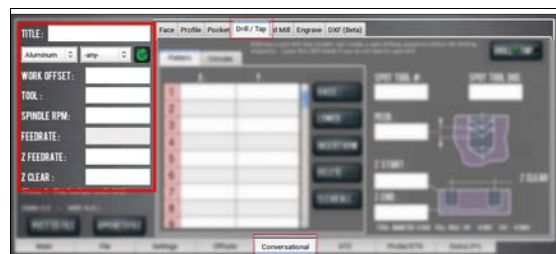


Figure 7-20: Drill tab on the Conversational tab.

2. From the **Drill** tab, select the **Pattern** tab.
3. In the **Location** table, select the row, then select the cell to edit.
Holes indicated in the **Location** table are completed from top to bottom.
4. Go to "Create a Drilling Sequence" (on the next page).

Tips

- To rearrange the row order, select a row, and then select **Raise** or **Lower**.
- To clear all values in the table, select **Clear All**.
- To make sure that the value is a valid number, leave the cell. If the value isn't a valid number, it's erased, and an error displays on the **Status** tab.

- Before posting the file, make sure that there are none of the following:
 - X values without a Y value
 - Y values without an X value
 - An empty row before the last row with values

About Drilling

A drilling program is used to create a series of holes in defined locations on a part.

Drilling in PathPilot

PathPilot has the following options to define the locations:

- **Pattern** Use X and Y locations to make a list of hole locations.
- **Circular** Use a circumference to make a specific pattern of evenly spaced holes.

For information on using conversational programming in PathPilot to drill holes, see "Create Hole Locations on a Part (Drill)" (on the previous page).

During a drilling routine, PathPilot uses one of the following canned G-code cycles, depending on the entries in each DRO field:

- **G81** Drill
- **G82** Drill with dwell
- **G83** Drill with peck



Note: The features can't be combined, because peck cancels dwell.

Create a Drilling Sequence

The Drill tab uses one of the following canned G8x cycles to drill a hole at each identified location:

- **G81** Drill
- **G82** Drill with dwell
- **G83** Drill with peck



Note: The features can't be combined, because peck cancels dwell.

Hole depth is usually defined as the full diameter portion of the hole, so you may need to consider the Z length from the drill point to the corner.

1. In the **Spot Tool #** DRO field, type the number of the spot drill.

If a valid tool number is used in the **Spot Tool #** DRO field, PathPilot makes a spot drilling sequence before the drilling sequence. In the spot drilling sequence, PathPilot uses the values indicated in the **Feedrate** DRO field, the **Spindle RPM** DRO field, and the **Z Clear** DRO field.



Note: To skip a spot drilling sequence, leave the **Spot Tool #** DRO field blank.

2. In the **Spot Tool DOC** DRO field, type the depth of cut for the spot drilling sequence.



Note: To skip a spot drilling sequence, leave the **Spot Tool DOC** DRO field blank.

3. In the **Peck** DRO field, type the distance for each peck.



Note: Any entry greater than **0** typed in the **Peck** DRO field replaces G81 with G83 in the G-code. For information, see Drilling and Tapping Reference.

4. Set the parameters for the hole in the Z range:
 - a. In the **Z Start** DRO field, type the location for the first Z pass.
 - b. In the **Z End** DRO field, type the location for the last Z pass.

Drilling Reference

PathPilot uses the following terms when drilling on a part in conversational programming:

- **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:
 - Positive or negative
 - Up to 90 degrees (or -90 degrees)
 A negative angle produces a clockwise rotation; a positive angle produces a counterclockwise rotation.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

- Rotates the pattern either clockwise or counterclockwise
For example, to create a hex pattern with flats on the top and bottom, use a value of 0 in the Start Angle DRO field. To create a hex pattern with flats on the left and right sides, use a value of 30 (or -30) in the Start Angle DRO field.

- Peck** The value isn't adjusted: the G83 routine feeds at the Z Feedrate, starting from Z Clear down to the Peck distance, then rapid retracts to Z Clear, and rapid returns to start the next Peck.



Note: The first and last peck will likely be shorter than the value typed into the Peck DRO field.

- Depending on the type of threads, do one of the following:

- For external threading (creating threads on a boss, for example), select **External**.
- For internal threading (creating threads inside a hole, for example), select **Internal**.

Create Threads on a Part

Using conversational programming, you can program PathPilot to take helical tool paths around a boss or inside a hole to create external or internal threads. For information, see "About Thread Milling" (on the next page).

Before You Begin

Before you begin, you must verify that you enter the program values considering the following:

- The auto-filled DRO field values assume a full-form threaded tool. If you're using a fine-point threaded tool to cut coarse threads, you must modify the root diameter to account for the tool's smaller nose radius.
- The value used in the **Z Clear** DRO field must be such that it can clear any blockages in the path between the end of one Z pass and the beginning of the next.

To create threads on a part:

- From the **Conversational** tab, select the **Thread Mill** tab.
- From the **Conversational DROs** group, set the parameters for the thread milling operation.



Figure 7-21: Conversational DROs on the Thread Mill tab.

4. Work through the program-specific DRO fields:
 - a. From the **Quick Reference** drop-down, select the thread size.
The **Threads/In** DRO field, **Pitch (In)** DRO field, **Major Dia.** DRO field, and **Minor Dia.** DRO field auto-fill.



Note: The threads listed follow the current unit setting: either inches (G20) or millimeters (G21).

- b. In the **Z Start** DRO field, type the location of the first Z pass (where you want the thread to start).
- c. In the **Z End** DRO field, type the location of the last Z pass (where you want the thread to end).



Note: The tool actually goes beyond the value entered in the **Z End** DRO field, due to the cutting tip width and the Z component of the compound feed angle and thread depth.

- d. In the **Depth of Cut** DRO field or the **Number of Passes** DRO field, type a value to represent the amount of material to remove in each helical pass. For information, see "Thread Milling Reference" (below).



Note: The **Depth of Cut** DRO field and the **Number of Passes** DRO field are linked — after you type a value in one and press **Enter** on the keyboard, the other is auto-filled.

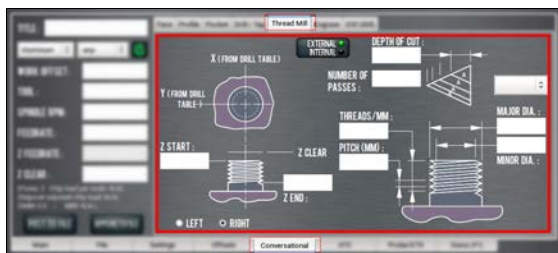


Figure 7-22: Program-specific DROs on the Thread Mill tab.

About Thread Milling

Thread milling is used to make helical tool paths on a part — either externally, like on a boss, or internally, like in a hole. The right-handed threads are based on pitch, diameter, and length.

Thread Milling in PathPilot

For information on using conversational programming in PathPilot to do thread milling, see "Create Threads on a Part" (on the previous page).

Thread Milling Reference

PathPilot uses the following terms when creating a thread mill on a part using conversational programming:

- **Major Dia. and Minor Dia.** Set 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 is the distance (the 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 degrees, keeping the cuts to one face of the tool.



Note: The Depth of Cut DRO field and the Number of Passes DRO field are linked — after you type a value in one and press **Enter** on the keyboard, the other is auto-filled.

- **Z Clear** The location the tool moves or retracts to when starting or ending a Z pass.

Create Text to Engrave on a Part

Using conversational programming, you can program PathPilot to engrave a single line of text on a part. For example, you could use this feature to engrave True Type stick or outline fonts into things like simple plaques, control panels, or data plates. For information, see "About Engraving" (on the next page).

To create an engraving sequence on a part:

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

1. From the **Conversational** tab, select the **Engrave** tab.
2. From the **Conversational DROs** group, set the parameters for the engraving operation.

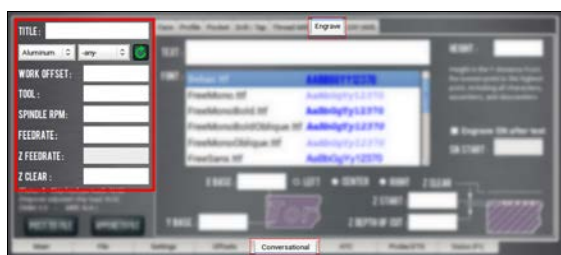


Figure 7-23: Conversational DROs on the Engrave tab.

3. Work through the program-specific DRO fields:
 - a. In the **Text** DRO field, type the desired text to engrave.
The text appears in the **Preview** window.
 - b. From the **Font** list, select the desired font for the engraved text.



Note: To search the **Font** list, select a font and begin typing the font's name. PathPilot scrolls to the result.

The **Preview** window updates to display the font selected.



Note: You can add True Type font files to the PathPilot controller. For information, see "Transfer Files to and From the Controller" (page 98).

- c. Use the **Alignment** radio buttons to select the desired alignment of the text: either **Left**, **Center**, or **Right**.
The **Preview** window updates to display the font selected.
- d. In the **Height** DRO field, type the distance in the Y direction from the top of the text to the bottom of the text.
PathPilot uses the height value, along with font data, to calculate scale for character paths in the G-code program.



Note: The value typed in the **Height** DRO field includes ascenders and descenders, but no the tool cutting diameter. For a more accurate value, subtract the diameter from the overall height.

- e. In the **X Base** DRO field and the **Y Base** DRO field, type the location of the left side of the first character's baseline.



Note: If any characters in the text have descenders, like **y** or **g**, they extend below the baseline.

- f. In the **Z Start** DRO field, type the location of the surface on which to engrave.
- g. In the **Z Depth of Cut** DRO field, type the desired depth the that the cutting tool is fed into the workpiece.
- h. In the **Z Clear** DRO field, type the value for the tool to move or retract to — in the **Z** direction — at the start and end of the engraving sequence and between characters.

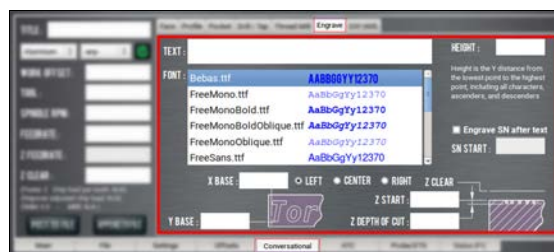


Figure 7-24: Program DROs on the Engrave tab.

About Engraving

Engraving is used to engrave a single line of text cut in a single horizontal pass along the X-axis of a part.

Import a DXF File

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. From the **Conversational** tab, select the **DXF** tab.

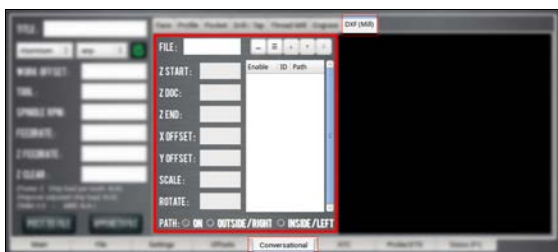


Figure 7-25: DXF tab on the Conversational tab.

2. Select the **File** DRO field.
The **File Selector** dialog box opens.
3. Select the .dxf file, and then select **Open**.
4. The shapes from the selected file are loaded into the **Preview** window.



Note: The .dxf file must already be transferred to the PathPilot controller. For information, see "Transfer Files to and From the Controller" (page 98).

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



EXAMPLE

If you type 1.0 in the **Scale** DRO field, the .dxf is scaled at 100%.

If you type 2.0 in the **Scale** DRO field, the .dxf is scaled at 200%.

7. In the **Rotate** DRO field, type the rotation angle in degrees.
The rotation angle is applied around the Z-axis of the drawing's origin.
8. Select one of the following to set the cutter compensation to be applied to the tool path:
 - **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:** The opposite of **Outside / Right**.

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.

Change the Layer or Shape Cut 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 is the same as the order of the enabled shapes in the tree view window and the cyan path in the **Preview** window.



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

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.

Working in the Preview Window

The **Preview** window uses the following colors:

- **Cyan** Selected paths
- **Gray** Disabled paths
- **White** Drawing path
- **Magenta** Cut path
- **Dark Cyan Stippled Line** Tool path between cuts
- The coordinates use the following colors:
 - **Red** X-axis
 - **Green** Y-axis
 - **Blue** Z-axis

7: PATHPILOT TOOLS AND FEATURES
7.2 Machine Settings and Accessories

7.2 MACHINE SETTINGS AND ACCESSORIES

Before running a G-code program, you must first make sure that the machine settings are properly configured.

7.2.1 Enable an Internet Connection 114
7.2.2 Change the Network Name 116
7.2.3 Change the Screen Orientation 116
7.2.4 Specify the Tool Change Method 117
7.2.5 Enable the On-Screen Keyboard 118
7.2.6 Enable Tooltips 118
7.2.7 Enable the USB M-Code I/O Interface Kit 119
7.2.8 Enable Feeds and Speeds Suggestions in Conversational Routines 119
7.2.9 Use a USB Camera 119

7.2.1 Enable an Internet Connection

If desired, you can enable an internet connection on your PathPilot controller. An internet connection allows you to receive automatic PathPilot updates and transfer files with Dropbox instead of a USB drive.

To enable an internet connection:

- 1. From the PathPilot interface, on the Status tab, select Internet.



Figure 7-26: Internet button on the Settings tab.

The Network Configuration dialog box displays.

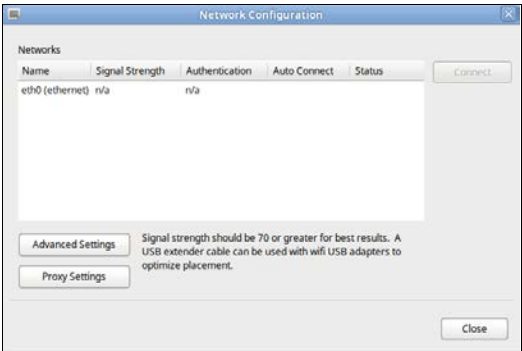


Figure 7-27: Network Configuration dialog box.

- 2. From the Network Configuration dialog box, in the Networks list, select the network you want to use. Then, select Connect.

Note: Wi-Fi connection signal strengths are indicated on a scale of 0 to 100, with 100 being the strongest. PathPilot continually refreshes the signal levels to help you find the best placement for your Wi-Fi network adapter. Ethernet connections are indicated by a prefix in the following format: eth[NUMBER]. For example, eth1.

The PathPilot operating system connects to the internet using the network you specified. It continues to detect and connect to the Wi-Fi network, even after power cycles.

- 3. Once connected, you can use the Dropbox and automatic updates features. Depending on what you want, see the following procedures:
• "Connect to Dropbox" (below)
• "Enable Automatic Updates" (on the next page)

Connect to Dropbox

Note: Dropbox requires an internet connection. If you haven't yet enabled it, go to "Enable an Internet Connection" (above).

If desired, you can connect your PathPilot controller to a Dropbox account to easily synchronize your G-code files, which eliminates the need to transfer them with a USB drive.

Note: Dropbox stops synchronizing once the PathPilot controller's internal drive has less than 500 MB of free space. To avoid this, we recommend that you organize your Dropbox account on a separate computer before you connect to Dropbox with PathPilot. Only store files in the top-level that you want synchronized to your PathPilot controller.

To connect to Dropbox:

- 1. From the PathPilot interface, on the Main tab, in the MDI Line DRO field, type ADMIN DROPBOX.

2. Select the **Enter** key.

The **Dropbox Configuration** application displays.

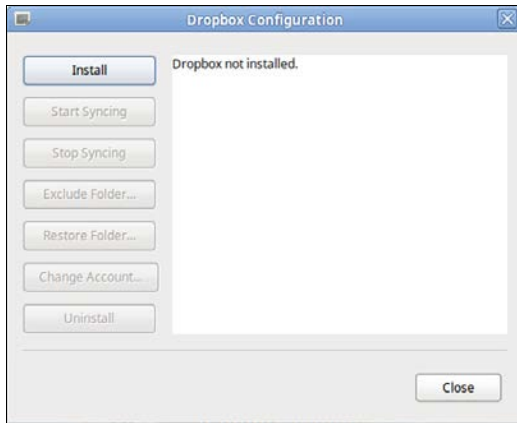


Figure 7-28: Dropbox Configuration application.

3. Select **Install**.

The **Dropbox Installation** dialog box displays.

4. Select **OK**.

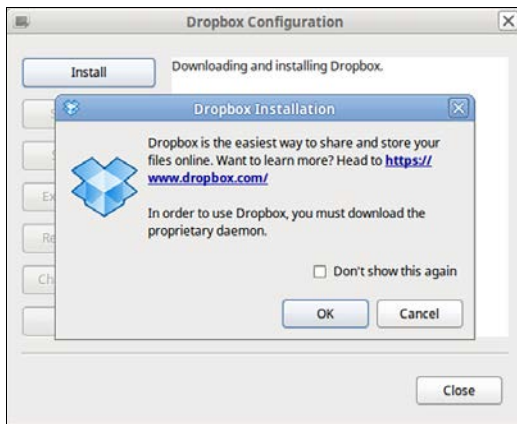


Figure 7-29: Dropbox Installation dialog box.

The Dropbox installation starts and continues for about a minute. When done, a web browser displays.

5. From the web browser, sign in or create a Dropbox account.
The PathPilot controller connects to the Dropbox account, creates a local **Dropbox** folder that is visible in **File** tab, and synchronizes the folder.
6. Because Dropbox stops synchronizing once the PathPilot controller's internal drive has less than 500 MB of free space, we recommend that you exclude large or unrelated folders from synchronization. Select **Exclude Folder....**

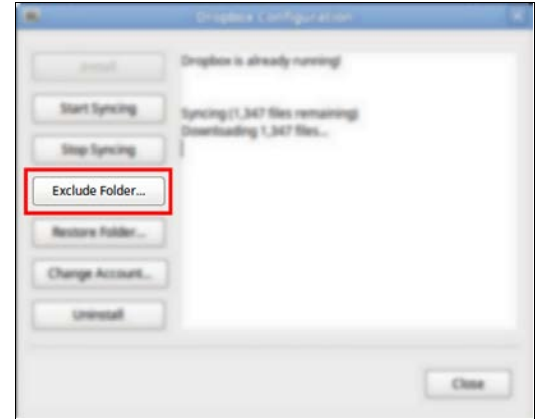


Figure 7-30: Exclude Folder button.

The **Add Folder to Excluded Set** dialog box displays.

7. In the **Add Folder to Excluded Set** dialog box, type the name of the folder to exclude.

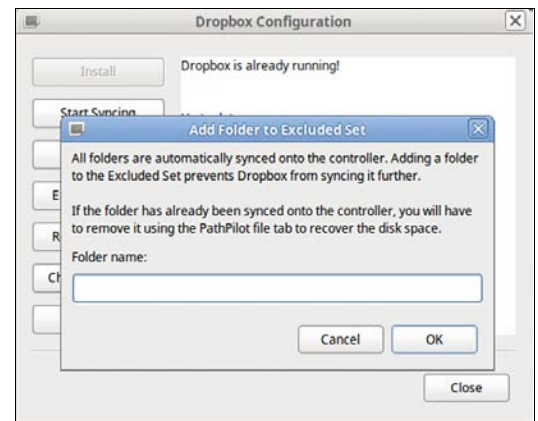


Figure 7-31: Add Folder to Excluded Set dialog box.



Note: You can only exclude folders, not individual files.

8. Select **OK**.
9. Select **Close**.

The PathPilot controller is now connected to Dropbox.

Enable Automatic Updates



Note: Automatic updates require an internet connection. If you haven't yet enabled it, go to "Enable an Internet Connection" (on the previous page).

If desired, you can enable automatic updates for PathPilot.
To enable automatic updates:

7: PATHPILOT TOOLS AND FEATURES
7.2 Machine Settings and Accessories

- 1. From the PathPilot interface, on the Status tab, select Update.
The Software Update dialog box displays.



Figure 7-32: Software Update dialog box.

- 2. From the Software Update dialog box, select the Check online daily for updates; confirmation required for download and installation checkbox.
3. Select Close.
When future updates are available, the Status tab displays a notification.

7.2.2 Change the Network Name

If you're connected to a network using either the Ethernet jack or the (optional) Wireless Network Adapter (PN 38207), the PathPilot controller appears on your network as network-attached storage. The default network name of the controller is TORMACHPCNC. To change the network name:

- 1. From the Network Name field, type a new network name.

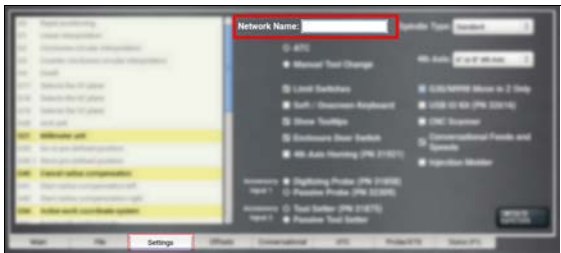


Figure 7-33: Network Name field on the Settings tab.

Note: The network name must be unique within your network.

- 2. Select the Enter key.
3. For the change to take effect, you must restart the controller.

7.2.3 Change the Screen Orientation

A vertical orientation for 1920 x 1080 monitors is supported in PathPilot v2.10.0 and later. For more information on the portrait layout, go to "About Portrait Screen Layout" (below). To change the screen orientation:

- 1. From the PathPilot interface, on the Settings tab, select Portrait from the Layout drop-down menu. Restart the controller.

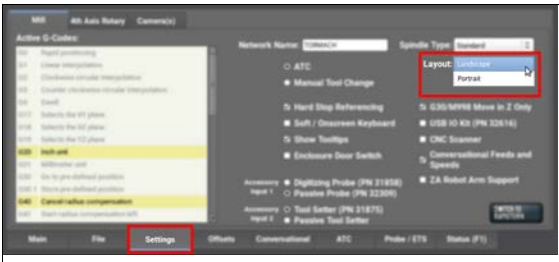


Figure 7-34: Layout drop-down menu on the Settings tab.

- 2. Rotate the monitor to the portrait orientation. You can rotate it either left or right, depending on what's easier for your setup.
3. While the controller is restarting, specify which direction you've rotated the monitor. Select Apply. If the result is unexpected, click Restore Previous Configuration on the confirmation dialog and choose a rotation direction again.

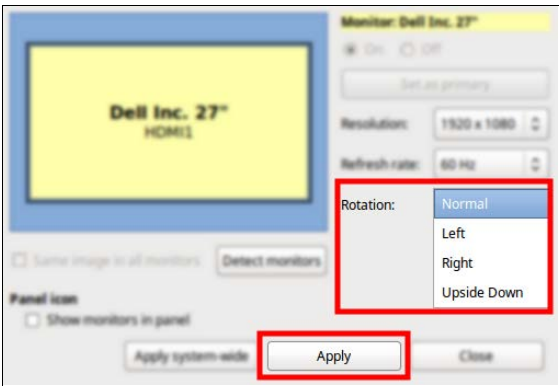


Figure 7-35: Monitor configuration dialog box.
The controller restarts in portrait layout.

About Portrait Screen Layout

Portrait layout provides some key advantages:

- A larger tool path window that's always visible at the top of the screen, regardless of which tab you have active.

7: PATHPILOT TOOLS AND FEATURES

7.2 Machine Settings and Accessories

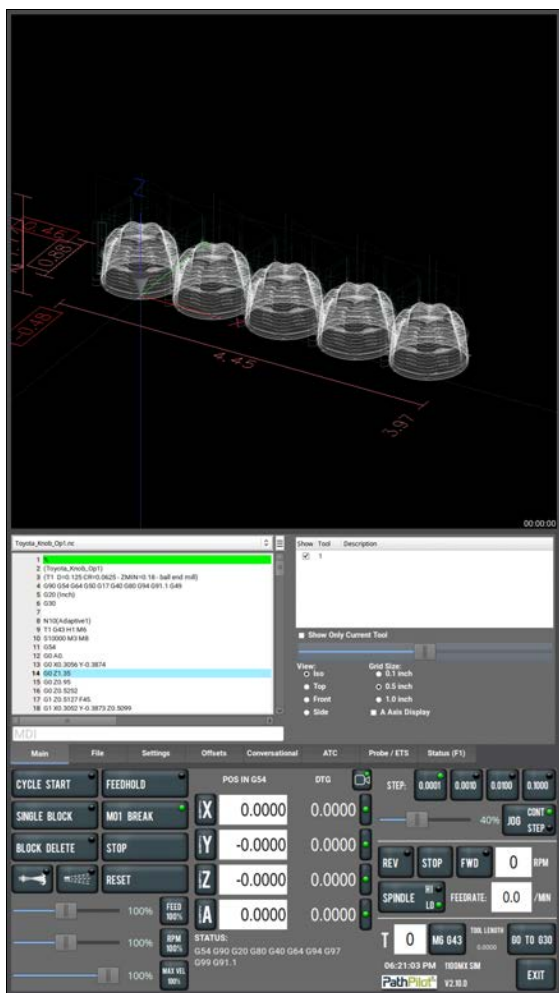


Figure 7-36: Tool Path window in portrait screen layout.

- A wider G-code window to more easily read the loaded G-code file and, if enabled, line numbers.
- The tool path window's view options are always visible for much easier access.
- When browsing G-code files using the File tab, file previews display on the top portion of the screen.

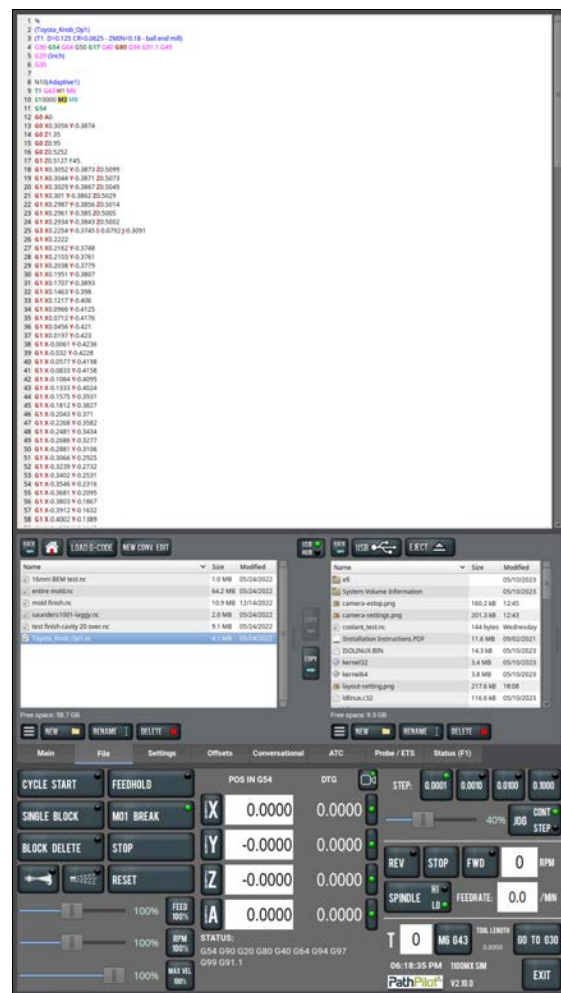


Figure 7-37: File tab G-code preview in portrait screen layout.

7.2.4 Specify the Tool Change Method

When PathPilot finds an **M06** command in a G-code program, it has different behaviors depending on the specified tool change method.

To specify the tool change method:

- From the **Settings** tab, select the appropriate tool change method for your machine.

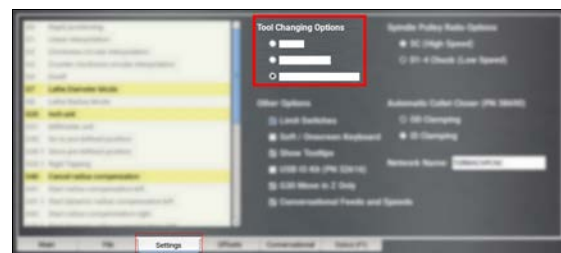


Figure 7-38: Tool change method options on the Settings tab.

7: PATHPILOT TOOLS AND FEATURES
7.2 Machine Settings and Accessories

About Automatic Tool Changes

If you're using an Automatic Tool Changer (ATC) to make tool changes, PathPilot does the following:

- 1. Looks for an ATC.
In the PathPilot interface, the ATC tab displays.
- 2. When PathPilot finds a Txx M06 command in the G-code program, the ATC loads the specified tool into the spindle.

About Manual Tool Changes

If you're making manual tool changes, PathPilot pauses when it finds an M06 command in the G-code program. While it's paused, PathPilot does the following:

- 1. The Cycle Start button flashes.
- 2. The Tool Path display shows the requested tool number.

7.2.5 Enable the On-Screen Keyboard

If you have an (optional) Touch Screen Kit (PN 35575), you can use a soft keyboard to type information in the PathPilot interface. For information, see "About Soft Keyboards" (below).

To enable and use the soft (on-screen) keyboard:

- 1. From the Settings tab, select Soft / On-Screen Keyboard.

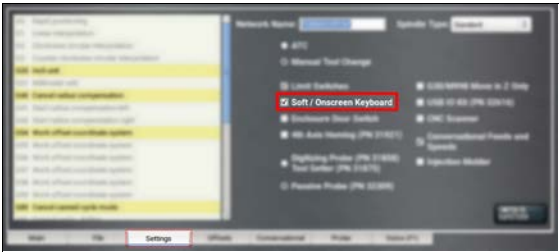
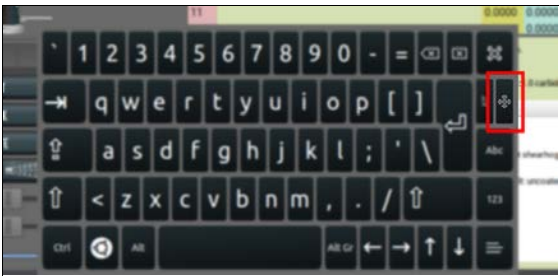


Figure 7-39: Settings tab.

- 2. To resize the keyboard, select a corner of the keyboard and drag.
- 3. To reposition the keyboard, select the Anchor key and drag the keyboard anywhere on the screen.



- 4. To close the keyboard, select the X key.



About Soft Keyboards

If you enabled a soft keyboard (on-screen keyboard) in the PathPilot interface to use with an optional touch screen or operator console, a keyboard opens when you select any field where keyboard input is required.

The keyboard displays a wide range of keys: both uppercase and lowercase, symbols, arrow keys, caps lock, backspace and delete, and more.



Figure 7-40: Soft (on-screen) keyboard.

7.2.6 Enable Tooltips

PathPilot displays expandable tooltips for many areas of the interface. Hovering over an item, like a DRO field or a button, displays helpful information about the item.

To enable or disable tooltips:

- 1. From the Settings tab, select or clear Show Tooltips.

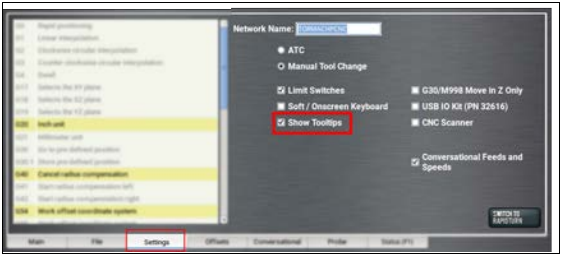


Figure 7-41: Show Tooltips checkbox.



Note: If you disable the tooltips, you can still display them for specific items. Hover over an area of the interface, and select the Shift key on the keyboard.

7.2.7 Enable the USB M-Code I/O Interface Kit

If you have a USB M-Code I/O Interface Kit (PN 32616), you must first enable it in the PathPilot interface.

To enable the USB M-Code I/O Interface Kit:

- From the **Settings** tab, select **USB IO Kit (PN 32616)**.

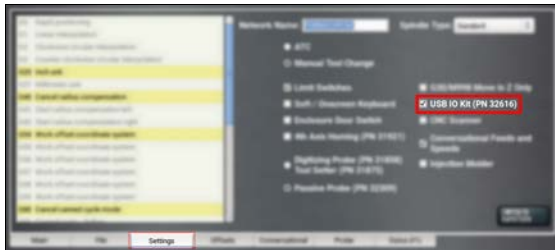


Figure 7-42: Settings tab.

7.2.8 Enable Feeds and Speeds Suggestions in Conversational Routines

You can use PathPilot to automatically calculate feeds and speeds. For more information, see "Use Feeds and Speeds Suggestions" (page 139).

- From the **Settings** tab, select **Conversational Feeds and Speeds**.

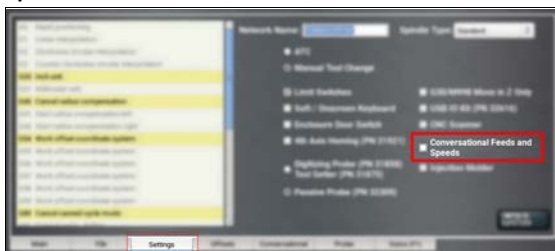


Figure 7-43: Settings tab.

7.2.9 Use a USB Camera

After plugging in the USB camera, navigate to the camera settings. From the PathPilot interface, in the **Settings** tab, open the **Camera(s)** tab. Identify the **Camera Status** read-only dialog box.



Figure 7-44: USB camera status.

As cameras are plugged in and unplugged, the **Camera Status** area is refreshed. To test compatibility of any USB camera, plug it in and watch the **Camera Status** area for the camera name and details.



Note: If a camera isn't shown after plugging it in or starting a video recording, it might require too much power from the USB ports on the controller. This is very likely when more than one camera is used. Try using a powered USB hub to add the camera(s).

When a USB camera is plugged in, it's analyzed for supported video and audio formats, frame sizes, and frame rates. If the camera supports it, PathPilot uses H.264 compression; otherwise, it uses Motion JPEG.

If the USB camera has a microphone, PathPilot records audio as well as video. The preferred format is compressed AAC, but uncompressed PCM is used as a fallback.

About USB Cameras

Recording video and audio from USB cameras is supported in PathPilot v2.10.0 and later. You can use up to four cameras simultaneously to record from different vantage points.



Note: All cameras are started and stopped at the same time — if you don't want a camera to record, you must unplug it.

USB cameras are compatible with all machine types, but older controllers may lack the processing power and memory needed for camera support. Controllers require 4GB of memory for camera functionality. Use the ADMIN MEMORY MDI command to verify the memory size of a controller.

You can purchase a Tormach USB Camera (PN 51240) with a metal case, mounting bracket, and 15-foot USB cable. Other USB cameras may work (see below), but do not include any technical support.

7: PATHPILOT TOOLS AND FEATURES
7.2 Machine Settings and Accessories

Manual Recording

To start or stop a manual recording, either:

- Use the controls in the Manual Recording area of the Camera(s) tab.
When a manual recording is stopped, a file save-as dialog appears prompting you for the file name base to use.



Figure 7-45: Manual recording controls.

- Select the Video Camera Recording button in the Persistent Controls section.
Whenever PathPilot is recording from a USB camera and/or the virtual screen camera, the LED on this button is green. If PathPilot is recording and the button is pressed:
- If a program is running and not paused at an M00/M01, the recording is aborted.
- If a program is not running, but the machine is moving, the recording is aborted.
- Otherwise, if a manual recording is in progress, it is stopped and a file save as dialog will appear. If an automatic e-stop loop recording is in progress, it is aborted since no e-stop occurred.



Figure 7-46: Video Camera Recording button.



Figure 7-47: Camera settings.

To take a picture (using all of the USB cameras at once):

1. Select Snapshot in the Manual Recording area of the Camera(s) tab.
The Main tab displays.
2. Review the camera images, which display on top of the Tool Path area. The camera images refresh every 0.5 seconds.
3. Align the cameras or adjust lighting to your preference, and then select the Shutter button.

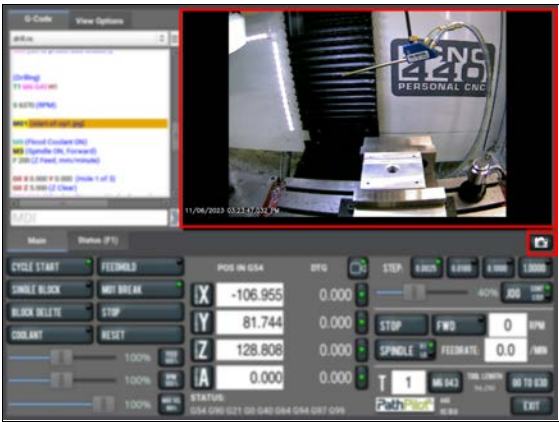


Figure 7-48: Example of taking a photo.

Automatic E-Stop Loop Recording ("Dashcam")

E-stop loop recording enables analysis of the previous 30 seconds after an E-stop. When enabled, recording is automatically started after reset.

To enable or disable the recording of emergency stops:

1. Toggle the Automatic e-stop loop recording checkbox in the Camera Settings area of the Camera(s) tab.

Note: This feature is enabled by default.

Automatic E-stop loop recording starts when the Reset button is selected. If you selected Video Camera Recording to abort a previous E-stop loop recording, select Reset to start it again.

To view E-stop videos:

1. A slight delay occurs after an E-stop while the video is saved to the **E-stop Videos** folder. Select the video file, and then select **Load G-Code** to view it.



Note: The E-Stop Videos folder is automatically monitored for internal drive space use. If the folder size grows beyond 5 GB, the oldest video files are automatically deleted until the folder size becomes less than 5 GB.

Review Video and Image Files

1. On the **File** tab, select the video or image file and select **Load G-Code**.

A video player application starts or the image preview is displayed.

Alternatively, you could transfer the video or image files to a Windows or macOS computer for review.

File Naming Convention

For manual and automatic E-stop recordings, the base file name for the recording has automatically chosen suffixes appended for each camera.

For example, if you stop a manual recording of two cameras, specify "Left Bracket Op1" as the name, and enabled screen recording, you'll see the following files:

File Name	Description of File
Left Bracket Op1_0.mp4	Camera 0 mp4 video file
Left Bracket Op1_0.log	Troubleshooting log for camera 0
Left Bracket Op1_1.mp4	Camera 1 mp4 video file
Left Bracket Op1_1.log	Troubleshooting log for camera 1
Left Bracket Op1_PP.mp4	PathPilot screen recording mp4 video file
Left Bracket Op1_PP.log	Troubleshooting log for screen recording

G-Code Commands

PathPilot supports three new M-codes to control cameras within G-code programs: M301, M302, and M303. Example use cases:

- Record only across each M01 stop where the operator needs to flip a workpiece or change a tool.
- Create short videos that focus on unique aspects of the program to reduce later video editing.
- Record USB IO integration operations with robots or other devices (pneumatic vises, etc.).
- Monitor progress on a workpiece by including M303 throughout the program.

File Naming Conventions

Recordings or pictures created by M301/M302/M303 have automatically generated file names, with the base file name taken from the running G-code file. Video files are saved alongside the G-code file. The suffix for each file uses a time stamp format. This makes it easy to distinguish multiple runs of the same G-code program.

For example, if engrave.nc is running and uses M301 and M302 to create one recording on a machine with one camera, and screen recording is enabled, you'll see the following files:

File Name	Description of File
engrave_2023-02-21_16_58_33_0.mp4	Camera 0 mp4 video file
engrave_2023-02-21_16_58_33_0.log	Troubleshooting log for camera 0
engrave_2023-02-21_16_58_33_PP.mp4	PathPilot screen recording mp4 video file
engrave_2023-02-21_16_58_33_PP.log	Troubleshooting log for screen recording
engrave_2023-02-21_17_43_22.jpg	Picture taken by a single M303 later in the program

Use M01 to Take Pictures

In addition to displaying information like pictures or messages during an M01 break, you can also use a USB camera (if installed) to take a picture.

To use M01 to take pictures:

7: PATHPILOT TOOLS AND FEATURES

7.2 Machine Settings and Accessories

1. Add M01 (`op1_setup.jpg`) into your G-code program.
2. Run the G-code program.
3. When PathPilot executes the M01 it looks to see if the comment contains a file name.
 - If there isn't a file name: The comment is shown as instructional text across the tool path.
 - If there is a file name, but the file doesn't exist yet and the extension is .jpg, .png, or .jpeg: The USB cameras are initialized and shown in the tool path display.
4. Select the **Shutter** button to take the picture and create the `op1_setup.jpg` file.

In future runs of the G-code program, **op1_setup.jpg** will display to the operator for instructional purposes on the workpiece.

For more information, see "Display Information and Capture Images During an M00 or M01 Break" (page 199).

7.3 SET UP G-CODE PROGRAMS

Before running a G-code program, you must first make sure that the machine is properly set up for the specific G-code program.

7.3.1 Use a Probe with PathPilot	123
7.3.2 Set Tool Length Offsets	125
7.3.3 Set Work Offsets	127
7.3.4 View Work Offsets	128
7.3.5 View Available G-Code Modes	128

7.3.1 Use a Probe with PathPilot

Use the Probe tab in the PathPilot interface to automate functions with a probe.

Set Up the Probe

Before using the functions on the Probe tab, you must first do the following:

1. Verify that tool number 99 (the probe tool) is in the spindle.
2. Disable the spindle to prevent any accidental spindle starts with the probe in the spindle.
3. Verify that the feed rate is appropriate for probing moves.

Note: All probing moves occur at a feed rate specified by the DRO fields on the Probe Setup tab.

4. Press the probe tip and make sure that, from the PathPilot interface, on the Probe tab, the Accessory Input light comes on.
This indicates that the probe polarity is correctly specified.
If the Accessory Input light does not come on, you must change the probe polarity setting. For information, see Enable a Probe.

Use a Probe to Find a Feature's Location

To find the location of a workpiece or vise in the current work offset coordinates:

1. From the PathPilot interface, on the **Probe** tab, select the **X/Y/Z Probe** tab.
2. Position the probe near the workpiece or vise.
3. One at a time, select **Find X+**, **Find X-**, **Find Y+**, **Find Y-**, or **Find Z-**.



Figure 7-49: Probe tab.

The axis is probed, and the location of the probed surface is displayed.

Use a Probe to Set Work Offset Zeroes

You can set the work offsets of a workpiece or vise jaw using a probe.

Set the X and Y Work Offset Zero on the Corner of a Feature

1. From the PathPilot interface, on the **Probe** tab, select the **X/Y/Z Probe** tab.
2. Position the probe so that it is below the surface of the feature and 1 in. away from the vice jaw corner in the **X** and **Y** directions.
3. Select **Find Corner, Set Work Origin**.



Figure 7-50: Probe tab.

The axes are probed, and the location of the probed surface is set as the current work offset's X/Y origin.



Note: Select **Change Corner** to change the corner on which to probe.

Set the Work Offset Zeroes on a Feature

1. From the PathPilot interface, on the **Probe** tab, select the **X/Y/Z Probe** tab.
2. Position the probe near the workpiece or vise.

7: PATHPILOT TOOLS AND FEATURES

7.3 Set Up G-Code Programs

3. One at a time, select **Probe X+**, **Set Work Origin**, **Probe X-**, **Set Work Origin**, **Probe Y+**, **Set Work Origin**, **Probe Y-**, **Set Work Origin**, or **Probe Z-**, **Set Work Origin**.



Figure 7-51: Probe tab.

The axis is probed, and the location of the probed surface is set as the current work offset's origin.

Use a Probe to Find the Center of a Feature

You can find the center of a pocket, slot, or boss on a part using a probe.

Find the Center of a Pocket

1. From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
2. Position the probe near the center of the pocket.
3. Select **Find Center, Set Work Origin** as shown in the following image.



Figure 7-52: Probe tab.

Find the Center of a Slot

1. From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
2. Position the probe near the center of the slot.
3. Depending on the slot, do one of the following:
 - To probe the slot in the **X** direction only, select **Find Center, Set Work Origin** as shown in the following image.

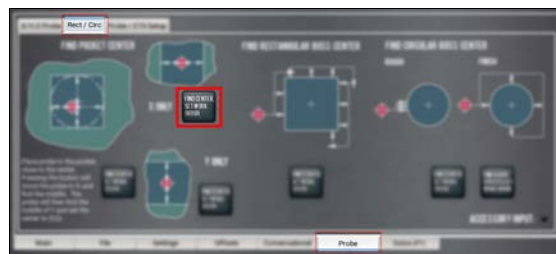


Figure 7-53: Probe tab.

- To probe the slot in the **Y** direction only, select **Find Center, Set Work Origin** as shown in the following image.

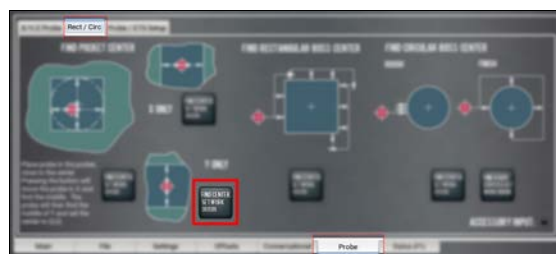


Figure 7-54: Probe tab.

Find the Center of a Rectangular Boss

1. From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
2. Position the probe below the top surface of the boss and on the left-hand side.
3. Select **Find Center, Set Work Origin** as shown in the following image.



Figure 7-55: Probe tab.

The probe moves around the edge of the workpiece to find the center.

Find the Center of a Circular Boss

1. From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
2. Position the probe below the top surface of the boss and on the left-hand side.

3. Select **Find Center, Set Work Origin** as shown in the following image.



Figure 7-56: Probe tab.

The probe moves around the workpiece three times to determine the approximate center of the curve, and then makes four additional move to confirm the center of the circle.

Find the Center Rotation of an A-Axis

1. From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
2. Position the probe directly above the A-axis center of rotation.
3. Select **Find A Axis Center & Set Work Origin**.

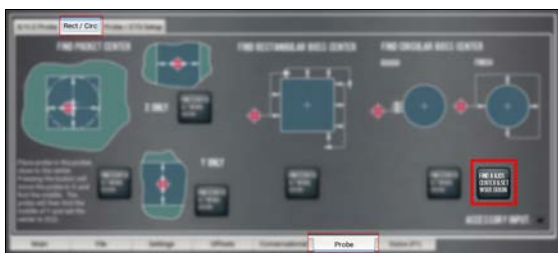


Figure 7-57: Probe tab.

The probe moves around the round workpiece mounted in the A-axis to find the center rotation of the A-axis.

7.3.2 Set Tool Length Offsets

Before running a G-code program, PathPilot must know the length of the tools that are required for the program. For more information on using tool length offsets, see "About Tool Offsets" (page 155).



Note: You can import a .csv file with tool length offset data. For information, see "Import and Export the Tool Table" (page 145).

To set tool length offsets:

1. Verify that the machine is powered on and out of reset.
2. Put a tool into a tool holder, and set it aside to measure. For information, see "Set Up Tooling" (page 165).

3. From the PathPilot interface, on the **Offsets** tab, verify that the **Tool** tab is selected.
4. Find the **Tool Table** window.

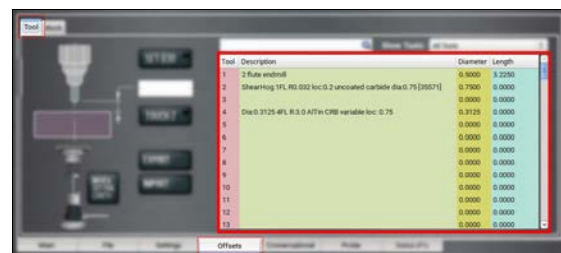


Figure 7-58: Tool Table window on the Offsets tab.

5. Depending on your workflow, you can measure your tools using any of the following methods:
 - **Use a Height Gauge** For information, see "Use a Height Gauge to Measure Tools" (page 157).
 - **Touch Off of a Known Reference Height** For information, see "Touch Off the Tool Length Offsets" (page 155).

About Tool Offsets

Tool offsets allow you to use various tools while still programming with respect to the workpiece. Tools can have different lengths (and, while using cutter radius compensation, different diameters).

The most common tool offset is the tool length offset: when you change tools, PathPilot must account for the difference in tool length. In CNC machines, the tool length offset is applied using a G43 command.

The tool length offset is the distance from the cutting edge of the tool to the shoulder of the tool holder. To make sure tool length offsets are consistent among your tools, we recommend you use Tormach Tooling System (TTS) tool holders. Set the tool length offset after placing a tool in the Tormach Tooling System (TTS) tool holder — each tool and its tool holder only need to be measured once.

Before you begin a G-code program, you must verify the lengths of the tools in the program, and make sure that the lengths agree with the tool length offsets set in PathPilot:

- Each time you change tools, you must apply a new tool length offset in PathPilot.

7: PATHPILOT TOOLS AND FEATURES
7.3 Set Up G-Code Programs

- Each time you replace a tool, you must remeasure its length, and apply a new tool length offset in PathPilot.

NOTICE! You must always verify that the physical length of a tool agrees with the tool length offset value set in PathPilot. If you don't, there's a risk that the tool length offset misrepresents the currently active tool in the spindle, which may result in a machine crash or damaged tooling, workpieces, or fixtures.

Touch Off the Tool Length Offsets

Touch off the tool length offsets by using a reference surface with a known height, which gives you a basis to measure any other tool lengths. Use any surface that is parallel (within 0.02 mm) to the machine table. For example:

- A 1-2-3 Block Set (PN 31950)
- Box parallel

There are two steps to touch off the tool offsets. Complete the following steps in the order listed:

Set a Known Reference Height 126
Measure Tools Using a Known Reference Height 126

Set a Known Reference Height

This procedure sets a new Z zero position for the currently selected work offset.

To set a known reference height:

- Identify a precision surface to use as a reference surface (like a 1-2-3 Block Set), and put it below the spindle on the machine table. Verify that there's a clear path from the spindle to the machine table.
- Verify that the drive dogs won't contact the reference surface before the end face of the spindle.
- Set a new, unused work offset (like G55). From the PathPilot interface, on the Main tab, in the MDI Line DRO field, type a work offset. Then select the Enter key. For information, see "Set Work Offsets" (page 158).
- If there's already a tool in the spindle, remove it.
- From the PathPilot interface, in the Tool DRO field, type 0. Then select the Enter key.
- Slowly jog the Z-axis down (-Z) until it's 0.04 in. (1 mm) from the reference surface.

- Measure the thickness of a piece of paper, and put the paper on the reference surface. Note the thickness of the paper for later.
- While moving the paper back-and-forth across the reference surface, slowly step the Z-axis down (-Z) until you feel a light pull on the piece of paper. This indicates that the paper is contacting the end face of the spindle.

Note: It's easier to use step jogging for this task. For information on step jogging, see "About Step Jogging" (page 150).

- From the PathPilot interface, in the Z-axis work offset DRO field, type the thickness of the piece of paper. Then select the Enter key.



Figure 7-59: Z-axis work offset DRO field.

The reference surface is now set as the Z zero position in the current coordinate system.

- To set the tool length offset, go to "Measure Tools Using a Known Reference Height" (page 170).

Measure Tools Using a Known Reference Height

This procedure sets the tool length offset using a known reference height. If you have not yet done so, you must first set the Z zero position; go to Set a Known Reference Height. To measure tools using a known reference height:

- Verify that the reference surface is still on the machine table with the piece of paper.
- From the PathPilot interface, on the Offsets tab, find an unused tool number in the Tool Table window. Then, type a description for the tool you're measuring.
- Put the tool holder into the spindle.
- From the PathPilot interface, in the Tool DRO field, type the number of the tool. Then select the Enter key.



Figure 7-60: Tool DRO field.

5. Slowly jog the Z-axis down (-Z) until it is 0.04 in. (1 mm) from the reference surface.
6. Continue to slowly jog the Z-axis while slowly moving the piece of paper back-and-forth on the reference surface.
7. Stop jogging the Z-axis when you feel a light pull on the piece of paper, which indicates that it is in contact with the tool.
8. From the PathPilot interface, on the **Offsets** tab, in the **Tool Table**, select the tool for which you previously wrote a description.
9. In the **Touch Z** DRO field, type the thickness of the piece of paper. Then select the **Enter** key.

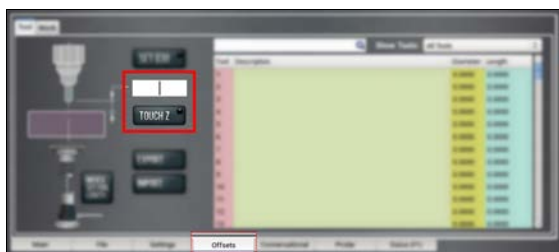


Figure 7-61: Touch Z DRO field and button.

10. Select **Touch Z**.
The length of the tool is stored in the **Tool Table** window.
11. From the **Tool Table** window, in the **Length** column, verify that the length of the tool is correct.
12. In the **Diameter** column, type the diameter of the tool. Then select the **Enter** key.
13. Jog the Z-axis up (+Z).
You've completed the procedure to measure a tool length offset. Repeat this procedure for any remaining tooling you have. Once you're done adding tool length offsets, switch back to your work coordinate system.

Use a Height Gauge to Measure Tools

1. Before measuring a tool, you must first zero the height gauge using a precision surface, like a Granite Surface Plate (PN 31713): Move the height gauge to the top of the Granite Surface Plate.
2. Press the **Zero** button on the touch trigger.
The height gauge is zeroed.
3. Insert a tool holder into the Granite Surface Plate.
4. From the PathPilot interface, on the **Offsets** tab, select the **Tool** tab.

5. In the **Tool Table** window, type a description for the tool.
6. Move the height gauge on the Tormach Tool Assistant Set to the end of the tool's cutting edge.
7. Write down the measured tool length, and then type that value into the **Length** column on the **Tool Table** window. Then select the **Enter** key.

7.3.3 Set Work Offsets

To set the current axis location to zero in the active work coordinate system:

- Select **Zero [Axis]**.

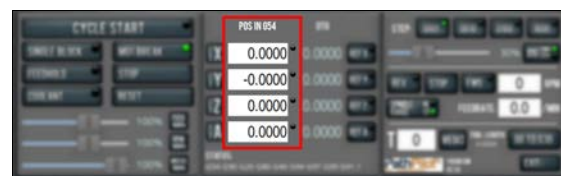


Figure 7-62: Work Offset DRO fields.

To change work offsets:

1. On the **Main** tab, in the **MDI Line** DRO field, type the new work offset to activate (for example, G55). Then select the **Enter** key.
2. The new work offset displays in the following locations in the PathPilot interface:
 - The **Status** read-only DRO field.
 - Above the **Work Offset** DRO fields.



Figure 7-63: Work offset indicated in the PathPilot interface.

Note: The values in the **Work Offset** DRO fields update to indicate the new location of each axis in the new work offset.

For more information on using work offsets, see "About Work Offsets" (page 168).

About Work Offsets

Work offsets allow you to think in terms of X, Y, and Z coordinates with respect to the part, rather than thinking of them with respect to the machine position. This means that you can jog the machine to an arbitrary location (like the end of a workpiece) and call that location zero.

7: PATHPILOT TOOLS AND FEATURES

7.3 Set Up G-Code Programs

You can save up to 500 work offsets in PathPilot. The naming structure varies based on the offset number, as detailed in the following table.

Work Offset Naming		
Offsets 1-9 (Use either name)		
Offset	Extended Name	Name
1	G54.1 P1	G54
2	G54.1 P2	G55
3	G54.1 P3	G56
4	G54.1 P4	G57
5	G54.1 P5	G58
6	G54.1 P6	G59
7	G54.1 P7	G59.1
8	G54.1 P8	G59.2
9	G54.1 P9	G59.3
Offsets 10-500 (Use extended name)		
Offset	Extended Name	Name
10	G54.1 P10	Not used
11	G54.1 P11	Not used
...		
499	G54.1 P499	Not used
500	G54.1 P500	Not used

7.3.4 View Work Offsets

To view the current work offset:

- From the **Offsets** tab, on the **Work** tab, identify the **Work Offsets Table** window.

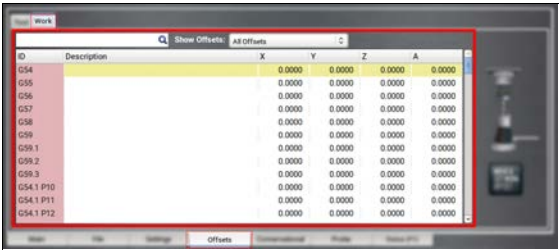


Figure 7-64: Work Offsets Table window.

The active work offset is highlighted.

To change the current work offset, go to "Set Work Offsets" (page 158).

7.3.5 View Available G-Code Modes

The G-Code Description window shows a list of all available G-code modes.

To view available G-code modes:

- From the **Settings** tab, find the **G-Code Description** window.

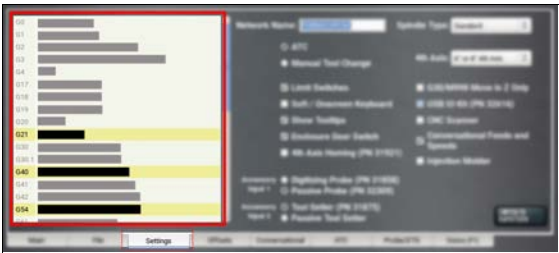


Figure 7-65: G-code Description window on the Settings tab.

7.4 RUN G-CODE PROGRAMS

While running a G-code program, use the following controls:

7.4.1 Bring the Machine Out of Reset	129
7.4.2 Disable Limit Switches	129
7.4.3 View the Active Axis to Jog	129
7.4.4 Jog the Machine	129
7.4.5 View the Current Machine Position	130
7.4.6 Reference the Machine	131
7.4.7 Start a Program	131
7.4.8 Stop Machine Motion	131
7.4.9 Operate the Coolant Pump	131
7.4.10 View the Active G-Code Modes	132
7.4.11 View the Distance to Go	132

7.4.1 Bring the Machine Out of Reset

- Select **Reset**.



Figure 7-66: Reset button.

For more information on reset mode, see "About Reset Mode" (below).

About Reset Mode

When the machine is first powered on, or after an emergency stop, the Reset button flashes. When you select the flashing Reset button, PathPilot verifies communication to the machine and does the following activities:

- Brings the machine out of an emergency stop condition
- Clears alarms
- Clears the tool path backplot
- Resets all modal G-codes to their normal state
- Rewinds the currently loaded G-code program
- Stops machine motion, but is **not** a replacement for the Emergency Stop button

You can select the Reset button any time while the machine is on.

7.4.2 Disable Limit Switches

To provide a temporary workaround for a malfunctioning limit switch circuit, you can disable the limit switches. For information, see About Limit Switches.



Note: By default, the Limit Switches checkbox is selected.

To disable limit switches:

1. From the **Settings** tab, clear the **Limit Switches** checkbox.

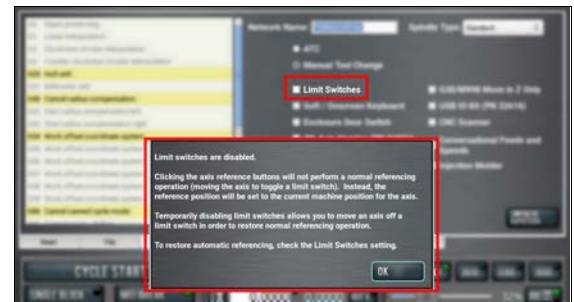


Figure 7-67: Limit Switches checkbox on the Status tab.

2. Select **OK**.

The machine completes a unique referencing procedure after selecting the axis reference buttons: rather than moving each axis to the end of its travel, the reference position is set as the machine's current position.



Tip! This is useful for troubleshooting, because you're now able to move the axis.

7.4.3 View the Active Axis to Jog

To find which axis is active while jogging your machine:

- Identify the light next to the **Work Offset** DRO fields.



Figure 7-68: Work Offset DRO fields.

For information, see "Jog the Machine" (page 150).

7.4.4 Jog the Machine

To switch between jogging modes:

- From the **Manual Control** area, in the **Jog** group, select **Jog**.
PathPilot toggles between continuous velocity mode and step mode.

7: PATHPILOT TOOLS AND FEATURES

7.4 Run G-Code Programs



Figure 7-69: Jog button.

When the **Cont** green light is on, continuous velocity mode is selected.

When the **Step** green light is on, step mode is selected.

To use continuous velocity mode:

- Set the velocity: drag the **Jog Speed** slider.



Figure 7-70: Jog Speed slider.

For more information on continuous velocity mode, see "About Continuous Velocity Jogging" (page 150).

To use step mode, select the step size. Do one of the following, depending on your accessories:

- In the **Manual Control Area**, in the **Jog** group, select the step size.
The **Step** button's light comes on, indicating which step size is active.



Figure 7-71: Step buttons (in G20 mode).

- On the (optional) Jog Shuttle, press the Step button to toggle the currently selected step size.
In the PathPilot interface, the **Step** button's light comes on, indicating which step size is active.

For more information on step mode, see "About Step Jogging" (page 150).

Jog in Continuous Velocity Mode

In continuous mode, the machine jogs at a continuous velocity. To select continuous velocity mode:

- In the **Manual Control** area, select **Jog**.



Figure 7-72: Continuous velocity jogging controls.

When the **Cont** green light is on, continuous velocity mode is selected.

When the **Step** green light is on, step mode is selected.

To set the velocity:

- Drag the **Jog Speed** slider.



Figure 7-73: Jog Speed slider.

Jog in Step Mode

In step mode, the machine jogs in steps, which range based on the programming mode you're using:

- **Imperial (G20) Mode** 0.0001 in. to 0.1000 in.
- **Metric (G21) Mode** 0.01 mm to 10 mm

To select the step size:

- In the **Manual Control Area**, select the step size.
The **Step** button's light comes on, indicating which step size is active.



Figure 7-74: Step buttons (in G20 mode).

7.4.5 View the Current Machine Position

- Identify the **Work Offset** DRO fields.

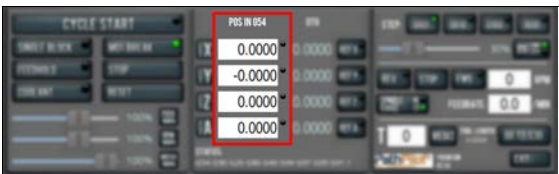


Figure 7-75: Work Offset DRO fields.

The position is expressed by the currently active work offset coordinate system (like G54 or G55).

When the machine isn't moving, you can edit the DRO fields. For more information on setting work offsets, go to "Set Work Offsets" (page 158).

7.4.6 Reference the Machine

1. Verify that the machine can freely move to its reference position (at the ends of travel).
2. To verify that the tooling is clear of any possible obstructions, reference the Z-axis before referencing the other axes: from the PathPilot interface, select **Ref Z**.



Figure 7-76: Reference buttons.

3. Once the spindle is clear of any possible obstructions, continue referencing all axes.



Note: You can select the buttons one after another. Once the machine references one axis, it'll move on to the next.

After each axis is referenced, its button light comes on. For more information on referencing the machine, see "About Referencing" (page 166).

About Referencing

You must reference the machine to establish a known position for PathPilot. The position that's set while referencing the machine is the origin of the machine coordinate system. Without referencing the machine, PathPilot won't know the current position of the machine axes.

You must reference the machine at the following times:

- After you power on the machine
- After you push in the Emergency Stop button
- Before running a G-code program
- Before using MDI commands
- Before setting work or tool offsets
- After a collision or an axis stall/fault

When referencing, the machine moves each axis to the end of its travel. The machine stops at the limit switch, which sets the axis' reference position.

7.4.7 Start a Program

- From the PathPilot interface, in the **Main** tab, select **Cycle Start**.



Figure 7-77: Cycle Start button.

For more information on starting a program, see "About Cycle Start" (below).

If you can't start a program, go to "Cycle Start Reference" (below).

About Cycle Start

While a program is running, the Cycle Start button's light is on. The Cycle Start button's light flashes if motion is paused during the program. The following modes may pause motion during a program:

- Single block
- Feed hold
- M01 break

If machine motion pauses a single block, feed hold, or M01 break, the Cycle Start button flashes until it's selected again.

Cycle Start Reference

The Cycle Start button doesn't operate if you select it:

- While you're not in the Main tab. For information, see "Main Tab" (page 89).
- Before you've loaded a G-code program. For information, see "Load G-Code" (page 153).
- Before referencing the machine. For information, see "Reference the Machine" (page 149).

7.4.8 Stop Machine Motion

- From the **Program Control** area, select **Stop**.



Figure 7-78: Stop button.

7.4.9 Operate the Coolant Pump

To turn coolant on or off:

7: PATHPILOT TOOLS AND FEATURES

7.4 Run G-Code Programs

- Select **Coolant**.



Figure 7-79: Coolant button.

For more information on turning on and off coolant, see "About Coolant" (page 159).

About Coolant

In the PathPilot interface, the Coolant button controls the machine's coolant pump power outlet. The Coolant button's light shows the current state of the outlet: the light is on when the outlet has power; the light is off when the outlet does not have power.

Note: The Coolant button is equivalent to using an M08 (coolant on) or M09 (coolant off) command in the G-code program.

Use the Coolant button before or after a program is running, while a program is running, or while you are using manual data input (MDI) commands.

7.4.10 View the Active G-Code Modes

To find the currently active G-code modes and the currently active tool at a glance:

- Identify the **Status** read-only DRO field.



Figure 7-80: Status read-only DRO field.

For more information on G-code modes, go to "View Available G-Code Modes" (page 128).

7.4.11 View the Distance to Go

To view the distance to go:

- Identify the **DTG** read-only DRO fields.



Figure 7-81: DTG read-only DRO fields.

The value is the remaining distance in any programmed move.

For more information on using the **DTG** read-only DRO fields, see "About Distance to Go" (below).

About Distance to Go

While a program is running, the DTG read-only DRO fields show the remaining distance in each move. After using the feed hold function or the maxvel override function, look at the distance to go. This read-only DRO field is useful to prove out a part program.

7.5 CONTROL G-CODE PROGRAMS

If necessary, use the following controls to add to your G-code program:

7.5.1 Use the Feed Hold Function	133
7.5.2 Use the Feed Rate Override Function	133
7.5.3 Use M01 Break Mode	134
7.5.4 Use the Maxvel Override Function	134
7.5.5 Use Single Block Mode	135
7.5.6 Use the Spindle Override Function	135
7.5.7 Change the Feed Rate	136
7.5.8 Change the Spindle Speed	136
7.5.9 Change the Tool Number	136
7.5.10 Use a G30 Position	137
7.5.11 View the Tool Length	137
7.5.12 Manually Enter Commands	137
7.5.13 Copy Recently Entered Commands	139
7.5.14 Limit G30 Moves	139
7.5.15 Use Feeds and Speeds Suggestions	139

7.5.1 Use the Feed Hold Function

- Select **Feed Hold**.



Figure 7-82: Feed Hold button.



Tip! Use the **Spacebar** key to quickly activate the feed hold function.

For more information on using the feed hold function, see "About Feed Hold" (below).

About Feed Hold

When the feed hold function is active, the Feed Hold button's light is on.

The feed hold function pauses machine motion — aside from the spindle — and the Cycle Start button flashes. For information, see "About Cycle Start" (page 131).



Note: If the machine isn't moving, the feed hold function doesn't have an effect.

You can use the feed hold function either while a program is running or while you are using manual data input (MDI) commands. If the program is running a spindle-synchronized move, the feed hold function is delayed until the move is complete.

7.5.2 Use the Feed Rate Override Function

To use the feed rate override function:

- Using the **Feed Rate Override** slider, change the programmed feed rate by a specific percentage.



Figure 7-83: Feed Rate Override slider.



Note: Percentages range from 1-200%.

To remove the feed rate override function:

- Select **Feed 100%**.
The feed rate returns to 100% of its programmed value (it's no longer overridden).

For more information on the feed rate override function, see "About Feed Rate Override" (below).

About Feed Rate Override

You can use the feed rate override function while you're doing any of the following activities:

- Using manual data input (MDI) commands
- Jogging
- Running a program with G01, G02, or G03 commands

The feed rate override function does not affect G00 (rapid) commands. It's ignored if:

- The program is running a spindle-synchronized move
- An M48 (disable feed and speed overrides) command is used

To indicate lack of motion or unusual levels, the slider turns yellow when it's either at 0% or above 100%.

The Feed Rate Override slider and Feed 100% button work similarly to the spindle override controls. They affect the commanded feed rate by a percentage from 1-200%. The feed rate override works for MDI, jogging, and G-code program G01/G02/G03 moves. The override has no effect on G00 (rapid) moves.

7: PATHPILOT TOOLS AND FEATURES

7.5 Control G-Code Programs

7.5.3 Use M01 Break Mode

- Select **M01 Break**.



Figure 7-84: M01 Break button.

For more information on using M01 break mode, see "About M01 Break" (below).

About M01 Break

When the M01 break mode is active, the M01 Break button's light is on. When the M01 break mode is inactive, the M01 Break button's light is off.

M01 break mode enables any M01 (optional stop) commands that are programmed in the G-code file. You can turn M01 break mode on or off either before starting a program or while a program is running.

- **When M01 Break is Active** Machine motion stops after PathPilot reaches an M01 command, and the Cycle Start button flashes. For information, see "About Cycle Start" (page 131).
- **When M01 Break is Inactive** PathPilot ignores all programmed M01 commands.

Display Information and Capture Images During an M00 or M01 Break

Display Information with Images

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 display when the program reaches the M00 or M01 break.

To display an image during an M00 or M01 break:

1. Move an image file with a .jpg or .png extension to the PathPilot controller in one of the following locations:
 - In the same folder as the G-code program
 - In an images folder within the G-code program's folder
 - In an images folder within the home directory
2. Program an M00 or M01 break, and, using parentheses, type the full file name of the image (including its extension).



EXAMPLE

M01 (Op1_Setup.jpg) displays the image file on the Tool Path display.

3. The image file displays on the Tool Path display.

Display Information with Text

To display a message on the Tool Path display:

1. Program an M00 or M01 break, and, using parentheses, type a message that you'd like to display on the screen.



EXAMPLE

M01 (Check coolant nozzles are pointed correctly) displays *Check coolant nozzles are pointed correctly* across the bottom of the Tool Path display.

2. The message displays on the Tool Path display.

Capture Images with a USB Camera

In addition to displaying information like pictures or messages during an M01 break, you can also use a USB camera (if installed) to take a picture.

To use M01 to take pictures:

1. Add **M01** (op1_setup.jpg) into your G-code program.
2. Run the G-code program.
3. When PathPilot executes the M01 it looks to see if the comment contains a file name.
 - If there isn't a file name: The comment is shown as instructional text across the tool path.
 - If there is a file name, but the file doesn't exist yet and the extension is .jpg, .png, or .jpeg: The USB cameras are initialized and shown in the tool path display.
4. Select the **Shutter** button to take the picture and create the op1_setup.jpg file.
In future runs of the G-code program, **op1_setup.jpg** will display to the operator for instructional purposes on the workpiece.

For more information, see "Use a USB Camera" (page 119).

7.5.4 Use the Maxvel Override Function

To use the maxvel override function:

- Using the **Maxvel Override** slider, change the maximum velocity by a specified percentage.



Figure 7-85: Maxvel Override slider.

To remove the maxvel override function:

- Select **Maxvel 100%**.

For more information on using the maxvel override function, see "About Maxvel Override" (below).

About Maxvel Override

The maxvel override function affects G00 and G01 commands, and it's useful for:

- **Running a Program for the First Time** Drag the Maxvel Override slider to 0% to verify that all DRO fields look appropriate.
- **Safety** If you're running a spindle-synchronized move, a maxvel override isn't ignored. Verify that the maxvel override value allows the machine to use the programmed feed rate during spindle-synchronized moves. If it can't, the spindle-synchronized move won't produce the results you want.

To indicate lack of motion or unusual levels, the slider turns yellow when it's either at 0% or above 100%.

7.5.5 Use Single Block Mode

- Select **Single Block**.



Figure 7-86: Single Block button.

For more information on using single block mode, see "About Single Block" (below).

About Single Block

While single block mode is active, the Single Block button's light is on.

Single block mode runs one line of G-code at a time. After each line, motion is paused, and the Cycle Start button flashes.

For information, see "About Cycle Start" (page 131).

You can turn single block mode on or off either before starting a program or while a program is running. For information, see "Use Single Block Mode" (above).



Note: Single block mode ignores non-motion lines, like comment lines or blank lines.

7.5.6 Use the Spindle Override Function

To use the spindle override function:

- Using the **Spindle Override** slider, change the programmed spindle speed by a specific percentage.



Figure 7-87: Spindle Override slider.



Note: Percentages range from 1-200%.

To remove the spindle override function:

- Select **RPM 100%**.
The spindle speed returns to 100% of its programmed value (it's no longer overridden).

For more information on using the spindle override function, see "About Spindle Override" (below).

About Spindle Override

The spindle override function won't command the spindle to move past the maximum allowable speed. If the spindle isn't moving, the spindle override function is delayed until the next time spindle starts. The override doesn't drive the spindle past its maximum speed. It does affect the speed of a spindle command limited by a D word.

You can use the spindle override function while you're doing any of the following activities:

- Running a program
- Using manual data input (MDI) commands

The spindle override function is ignored in the following situations:

- If the program is running a spindle-synchronized move
- If an M48 (disable feed and speed overrides) command is used

To indicate lack of motion or unusual levels, the slider turns yellow when it's either at 0% or above 100%.

7: PATHPILOT TOOLS AND FEATURES

7.5 Control G-Code Programs

7.5.7 Change the Feed Rate

- In the **Feed Rate** DRO field, type in a feed rate. Then select the **Enter** key.



Figure 7-88: Feed Rate DRO field.

For information, see "About Feed Rates" (below).

About Feed Rates

A feed rate is the velocity at which the workpiece can be fed against the tool in the machine's spindle.

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

7.5.8 Change the Spindle Speed

- In the **Spindle RPM** DRO field, type in a spindle speed. Then select the **Enter** key.



Figure 7-89: Spindle RPM DRO field.

For information, see "About Spindle Controls" (below).

About Spindle Controls

A spindle speed is the rate at which the spindle rotates.

Button	G-Code	Use to...
FWD	M03	Start the spindle clockwise at the RPM specified in the Spindle RPM DRO field.
Stop	M05	Stop the spindle.

The FWD button and the Spindle RPM field don't operate if selected when:

- A G-code program is running.
- Using manual data input (MDI) commands.

Spindle Controls Reference

The spindle speed is measured in revolutions per minute (RPM).

Two speed ranges are available:

- **Low** 300 rpm to 4500 rpm
- **High** 650 rpm to 10,000 rpm

Use the low speed range when you're using larger cutting tools, or machining materials with lower surface speeds (like steel or stainless steel).

Use the high speed range when you're using smaller cutting tools, or machining materials with higher surface speeds (like aluminum, brass, or plastic).

To move your spindle speed range from one to the other, see "Change the Spindle Speed Range" (page 152).

7.5.9 Change the Tool Number

The Tool DRO field shows the tool currently in the spindle.



Figure 7-90: Tool DRO field.

To change the tool number (and apply its tool length offset):

1. In the **Tool** DRO field, type a number (the valid range is from 0-1000). Then select the **Enter** key.



Note: You can also select **M6 G43**. For information, see "About M6 G43" (below).

About M6 G43

The M6 G43 button is a shortcut used to do the following:

- Change the number of the currently-loaded tool in the spindle to the number typed in the Tool DRO field. This is the equivalent of an M06 command.
- Apply the tool length offset for that tool typed in the Tool DRO field. For more information on tool length offsets, see "Set Tool Length Offsets" (page 155). This is the equivalent of a G43 command.

7.5.10 Use a G30 Position

The Go to G30 button moves the machine to a predefined G30 position. For information, see "About G30" (page 139).

To set a G30 position:

1. Jog the machine to the desired G30 position.
2. From the **Offsets** tab, select **Set G30**.

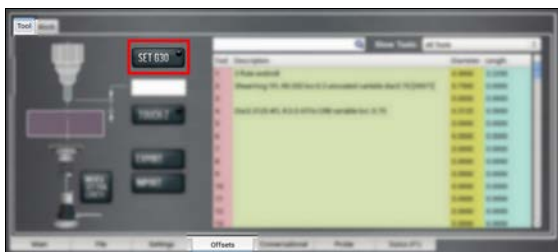


Figure 7-91: Set G30 button.

To go to a set G30 position:

- Do one of the following:
 - Use a **G30** command in a G-code program.
 - Select **Go To G30**.



Figure 7-92: Go to G30 button.



Note: The G30 position defaults to only moving the Z-axis.

About G30

A G30 command in a G-code program moves the machine to a preset position. For more information on setting a G30 position, see "Use a G30 Position" (above).

Use a G30 move to start a coordinated movement of the axes. You can limit the movement to only the Z-axis. For information, see "Limit G30 Moves" (page 139).



Tip! It's useful to program a G30 move right before a tool change so that the machine can jog to a safe tool change position.

7.5.11 View the Tool Length

- Identify the **Tool Length** read-only DRO field.



Figure 7-93: Tool Length DRO field.

If the tool offset matches the number of the tool in the **Tool DRO** field, the text is light blue on a gray background.

If the tool offset doesn't match the number of the tool in the **Tool DRO** field, the text is orange on a red background.

7.5.12 Manually Enter Commands

You can send G-code commands directly to the machine by using the MDI Line DRO field. For information, see "About the MDI Line DRO Field" (on the next page).

To manually enter commands:

1. Select the **MDI Line DRO** field.

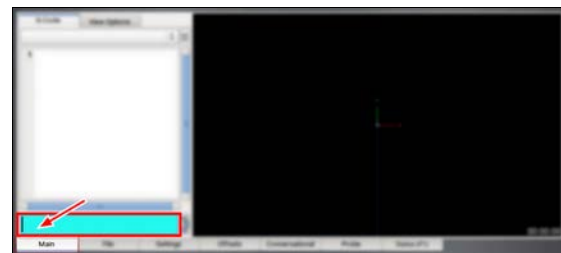


Figure 7-94: MDI Line DRO field.

The DRO field highlights.

2. Type the command.



Note: You can use the **Backspace**, **Delete**, **Left Arrow**, and **Right Arrow** keys to correct typing errors.

3. You must press the **Enter** key to execute the command. To abandon the command, press **Esc**.

7: PATHPILOT TOOLS AND FEATURES

7.5 Control G-Code Programs

About the MDI Line DRO Field

The MDI Line DRO field allows you to send commands (or, manual data input) directly to PathPilot. For information, see "Manually Enter Commands" (on the previous page). The MDI Line DRO field saves up to 100 of your most recent commands, which are saved after a power cycle. When you select the MDI Line DRO field, all keystrokes are used within the field — so, you can't jog the machine.

Admin Commands Reference

Use the following commands in PathPilot:

Admin Command	Use to...
ADMIN AUDIO	Customize the controller's audio device settings.
ADMIN CALC	Open the calculator.
ADMIN CLEAR	Clear the message history on the Status tab.
ADMIN CONFIG	Change the configuration of the PathPilot interface.
ADMIN CYCLECOUNTER	Control the cycle counters in the Tool Path display.
ADMIN DATE	Customize the controller's date and time.
ADMIN DISPLAY	Customize the controller's screen display.
ADMIN DROPBOX	Connect your controller to a Dropbox account for cloud file syncing.
ADMIN HELP	Review a list of available Admin commands.
ADMIN KEYBOARD	Customize the controller's keyboard layout.
ADMIN LOGDATA	Write the latest machine log data to a USB drive for technical support assistance.
ADMIN MEMORY	Determine how much total RAM is on your controller.

Admin Command	Use to...
ADMIN MOUSE	Change the mouse preferences, like pointer speed and right- or left-hand button mapping
ADMIN NETWORK	Configure a Wi-Fi network.
ADMIN OPENDOORMAXRPM	Change the maximum spindle speed with the spindle cabinet door open, when using the optional Enclosure Door Switch Kit.
ADMIN RESET_SOFT_LIMITS	Reset axis soft limits to machine defaults.
ADMIN SET_X_LIMIT	Set the X-axis soft limit.
ADMIN SET_Y_LIMIT	Set the Y-axis soft limit.
ADMIN SET_Z_LIMIT	Set the Z-axis soft limit.
ADMIN SETTINGS BACKUP	Create a backup of tool offset and fixture information to store externally.
ADMIN SETTINGS RESTORE	Restore tool offset and fixture information backup from an external location.
ADMIN SHOW_SOFT_LIMITS	Display the current axis soft limits on the Status tab.
ADMIN TOOLTIP DELAYMS	Set the milliseconds prior to displaying the tooltip (and then again for the expanded tooltip). The default is 1200 milliseconds.
ADMIN TOOLTIP MAXDISPLAYSEC	Limit the amount of time the expanded tooltip displays. The default is 15 seconds.
ADMIN TOUCHSCREEN	Adjust the touch screen calibration.
ADMIN VERSION	Display detailed version information on the Status tab.

7.5.13 Copy Recently Entered Commands

1. From the **MDI Line** DRO field, press either the **Up Arrow** key or the **Down Arrow** key.
The previously entered command displays.
2. You must press the **Enter** key to execute the command.
To abandon the command, press **Esc**.

For information, see "Manually Enter Commands" (page 137).

7.5.14 Limit G30 Moves

You can limit G30 moves so that only the Z-axis moves. For information, see "About G30" (below).

To limit G30 moves:

- From the **Settings** tab, select **G30/M998 Move in Z Only**.

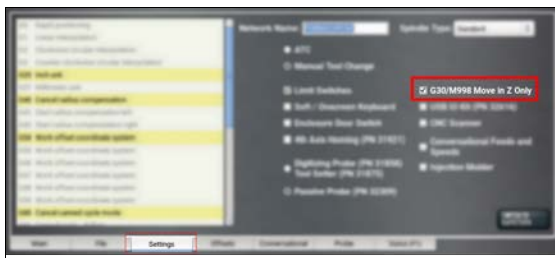


Figure 7-95: Settings tab.

About G30

A G30 command in a G-code program moves the machine to a preset position. For more information on setting a G30 position, see "Use a G30 Position" (page 137).

Use a G30 move to start a coordinated movement of the axes. You can limit the movement to only the Z-axis. For information, see "Limit G30 Moves" (above).



Tip! It's useful to program a G30 move right before a tool change so that the machine can jog to a safe tool change position.

7.5.15 Use Feeds and Speeds Suggestions



Note: Calculating feeds and speeds requires that PathPilot has relevant details about the tooling. If you haven't yet done so, go to "Create Tool Descriptions" (on the next page).

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.

1. If you haven't yet done so, enable the conversational feeds and speeds setting. From the **Settings** tab, select **Conversational Feeds and Speeds**.

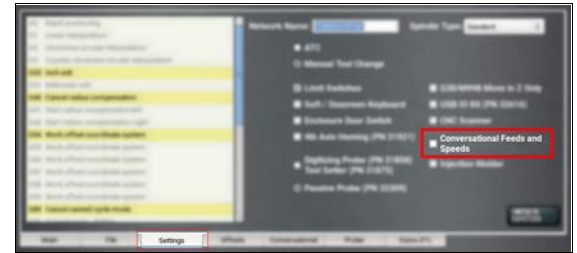


Figure 7-96: Settings tab.

2. From the **Conversational** tab, locate the **Material** dropdowns in the **Conversational DROs** group.



Figure 7-97: Feeds and speeds suggestions on the Conversational tab.

3. From the **Material** dropdown, select your material (like **Aluminum** or **Plastic**).
4. If required, from the **Sub-Type** dropdown, select the material sub-type (like **-any-** or **6061**).
5. In the **Tool** DRO field, type the assigned tool number.
6. Select **Refresh** (to the right of the **Sub-Type** dropdown). The following machining-related DRO fields are calculated:

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



Note: After PathPilot calculates values for the machining-related DRO fields, the background turns green.

7: PATHPILOT TOOLS AND FEATURES
7.5 Control G-Code Programs

7. (Optional) You can adjust the values in the calculated DRO fields. Adjusting the value in one of these DRO field doesn't change the value in the other machining-related DRO fields.

Note: Once you adjust the value in the DRO field, the background switches from green back to white. This helps you identify which DRO fields have suggested values (those with a green background), and which DRO fields have values you've supplied (white background).

Refresh DRO Field Values

The suggested values 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 field. 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. Changing your conversational operation requires you to refresh the feeds and speeds values.

When the feeds and speeds are no longer valid, the Refresh button turns green, and the machining-related DRO field backgrounds switch from green to white, as shown in the following image.



Figure 7-98: Refresh button on the Conversational tab.

Use Additional Provided Information

The following tips are displayed based on the calculations that PathPilot is performing:

- Chip Load Information Chip load — the amount of material removed per tooth — is based on the number of flutes, RPM, and feed rate.

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 premature tool wear.

- Cutting Speed Information Cutting speed is the speed that a given tooth (flute) on the cutter will be moving when it cuts through the material. All materials have a documented cutting speed. In imperial units, cutting speed is measured as surface feet per minute (SFM). In metric units, cutting speed is measured as surface meters per minute (SMM).
- Material Removal Information The material removal rate (MRR) indicates how much material is removed by the tool per minute while cutting. In imperial units, cutting speed is measured as cubic inches per minute. In metric units, cutting speed is measured as cubic centimeters per minute.

Enable Feeds and Speeds Suggestions in Conversational Routines

You can use PathPilot to automatically calculate feeds and speeds. For more information, see "Use Feeds and Speeds Suggestions" (on the previous page).

- From the Settings tab, select Conversational Feeds and Speeds.

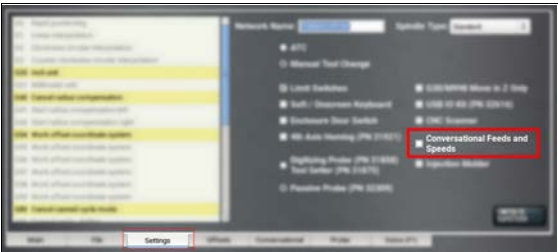


Figure 7-99: Settings tab.

Create Tool Descriptions

If desired, you can create tool descriptions in PathPilot. Detailed tool descriptions allow you to receive feeds and speeds suggestions in conversational programming. For information, see "Use Feeds and Speeds Suggestions" (on the previous page).

PathPilot uses keywords and patterns in the tool description to recognize tooling features. For information, see "Tool Keywords Reference" (below).

1. From the PathPilot interface, on the **Offsets** tab, identify the **Tool Table** window.
2. Select a blank line.
3. Type a description for the tool. Descriptions are not case sensitive.

Examples

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

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

 EXAMPLE

This description provides very little information, and PathPilot defaults to basic cutter features:

- ## Automatically Generate Tool Descriptions

 **Note:** If you don't know the part number, you can search for the tool at tormach.com.

2. Type the part number for the tool, like **35571**.

The screenshot shows the 'Tool' menu in the software. The 'Tool' menu is open, displaying a list of tools. The 'Tool' menu is highlighted in red. The 'Tool' menu is open, showing a list of tools. The 'Tool' menu is highlighted in red.

Figure 7-101: An automatically-generated tool description for a Tormach tool.

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

Tool Keywords Reference

PathPilot uses keywords and patterns in the tool description to recognize tooling features.

7: PATHPILOT TOOLS AND FEATURES

7.5 Control G-Code Programs

Item	Pattern	Example	Notes
type	drill, centerdrill, tap, ball, chamfer, spot, flat, taper, bullnose, lollipop, flycut, shearhog, drag, saw, indexable	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”	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	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	TIN, ZRN, TIB2	No coating is specified same as “uncoated.”
tool diameter	“diameter” or “dia” followed by a colon, followed by a decimal number	DIAMETER:.0341 , DIA:.750	—

Item	Pattern	Example	Notes
tool material	carbide, HSS, CoHSS, CRB, carb, diamond, DMND	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	R:.02, RADIUS:0.02	No radius is specified the same as a zero radius.

7.6 SYSTEM FILE MANAGEMENT

To keep the files on your system backed up and organized, use the following controls:

7.6.1 Manage System Files	143
7.6.2 Create Backup Files	143
7.6.3 Restore Backup Files	144
7.6.4 Import and Export the Tool Table	145

7.6.1 Manage System Files

Use the File tab to manage system files on the PathPilot controller. For information, see "About System Files" (below).

To manage system files:

- From the PathPilot interface, on the **File** tab, do any of the following from the **Controller Files** window:
 - Select a file, and then select **New Folder**, **Rename**, or **Delete**.
 - Select a file, and go to the **Options** menu. Then, select **Copy**, **Cut**, or **Paste**.

To navigate through the system files:

- Select **Back** or **Home**.

About System Files

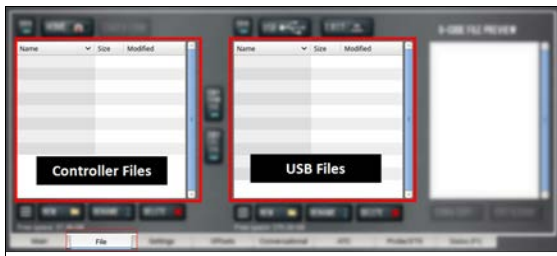


Figure 7-102: File tab.

PathPilot doesn't run G-code program files from a USB drive. You must first transfer files to the PathPilot controller. For information on transferring files, see "Transfer Files to and From the Controller" (page 98).

7.6.2 Create Backup Files

1. Insert a blank, formatted USB drive into the PathPilot controller.

Note: To prevent errors when backing up and restoring files, only use a blank, formatted USB drive. Do not use the PathPilot v2.0 Upgrade.

2. From the PathPilot interface, on the **Main** tab, in the **MDI Line DRO** field, type `ADMIN SETTINGS BACKUP`.

Then select the **Enter** key.

PathPilot generates a backup .zip file, and the **Admin Settings Backup** dialog box displays.

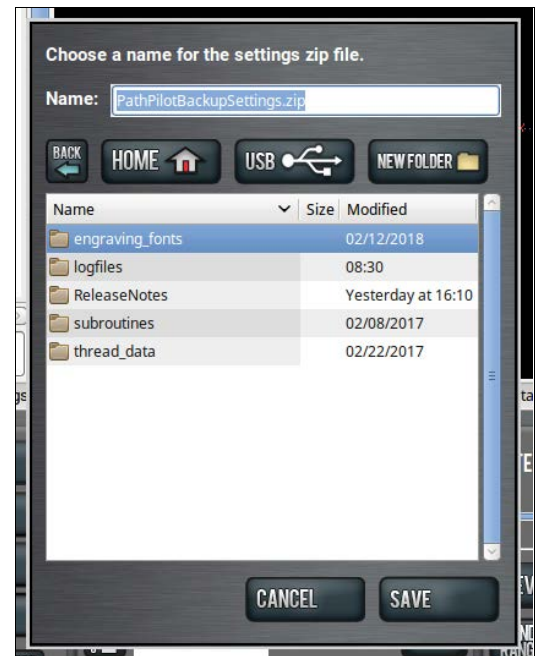


Figure 7-103: Admin Settings Backup dialog box.

3. From the **Admin Settings Backup** dialog box, specify where (on the PathPilot controller or on a USB drive) to save the backup .zip file.
 4. Select **Save**.
- The backup .zip file is saved in the location you specified in Step 3.

7: PATHPILOT TOOLS AND FEATURES

7.6 System File Management

5. If you saved the backup .zip file on the PathPilot controller, you must manually transfer it — along with other files you want to back up (like G-code programs) — to a USB drive. From the PathPilot interface, on the **File** tab, in the **Controller Files** window, select the backup .zip file and any other files you want to back up.



Figure 7-104: Controller Files window on the File tab.



Note: Files must have unique names. If they don't, PathPilot prompts you to overwrite or rename files, or cancel the file transfer.

6. To prevent errors, make sure you don't include the following folders:
 - logfiles
 - media
 - ReleaseNotes
 - subroutines
 - USB
7. Select **Copy to USB**.
The files are copied and display in the **USB Files** window.
8. Eject the USB drive from the PathPilot controller.
9. From the PathPilot interface, select **Exit**.
10. Verify that all files are properly saved: insert the USB drive on a device other than the PathPilot controller, and review the list of files on the USB drive.
11. (Optional) As an extra precaution, copy all the files onto the device.
12. Go to Install PathPilot v2.x.

About Backup Files

Make a regular backup of all tool offset and fixture information and machine settings stored on your PathPilot controller. Store the file externally to use if you replace your controller or restore it to factory settings.

7.6.3 Restore Backup Files

1. Insert the USB drive with your backup files into the PathPilot controller.
2. From the PathPilot interface, on the **Main** tab, in the **MDI Line DRO** field, type `ADMIN SETTINGS RESTORE`. Then select the **Enter** key.

The **Admin Settings Restore** dialog box displays.

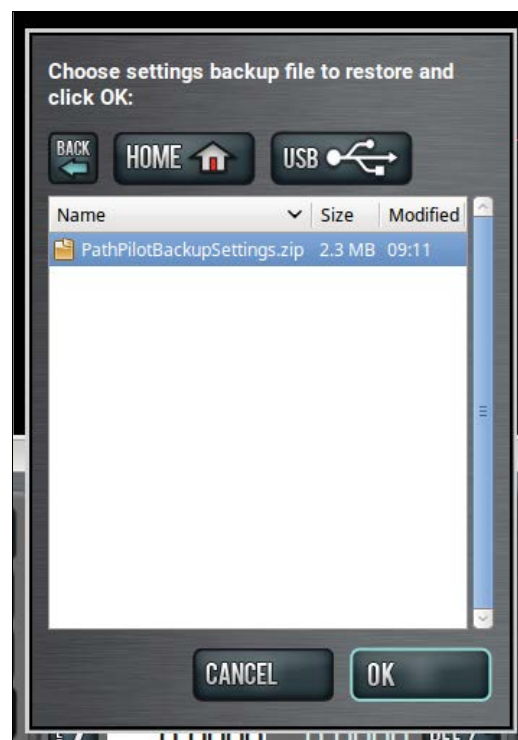


Figure 7-105: Admin Settings Restore dialog box.

3. From the **Admin Settings Restore** dialog box, navigate to the backup .zip file on the USB drive, and then select **OK**. The PathPilot operating system restores the backup, then restarts.

- If you backed up any other files onto the USB drive, you must manually transfer the files to the PathPilot controller. From the PathPilot interface, on the **File** tab, in the **USB Files** window, select the files you want to transfer.

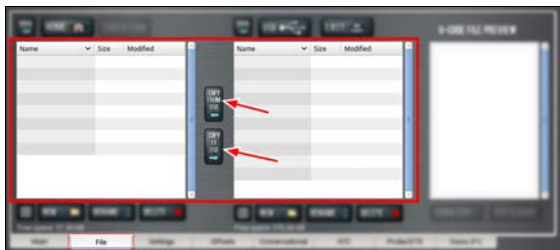


Figure 7-106: USB Files window on the File tab.



Note: To navigate backward, select **Back**. To navigate to the top level, select **USB**.

- From the **Controller Files** window, select the folder into which you want to copy the files.
- Select **Copy From USB**.
The files display in the **Controller Files** window.



Note: Files must have unique names. If they don't, PathPilot prompts you to overwrite or rename files, or cancel the file transfer.

- If desired, you can enable an Internet connection in the PathPilot controller, which allows you to use Dropbox with PathPilot, and receive automatic PathPilot updates. Go to the next section, "Enable an Internet Connection" (page 114).
If you don't want to use these features, you have completed installing and configuring PathPilot v2.x.

7.6.4 Import and Export the Tool Table

You can manage the tool table using an external .csv file.

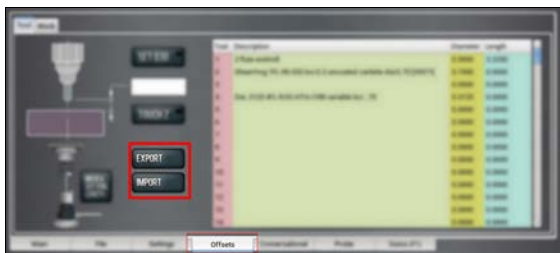


Figure 7-107: Export and Import buttons on the Offsets tab.

Import a .csv File

- Transfer the .csv file to a USB drive.
- Insert the USB drive into the PathPilot controller.
- Confirm that the PathPilot controller is on.
- From the **Offsets** tab, select **Import**.

The **Import** dialog box displays.

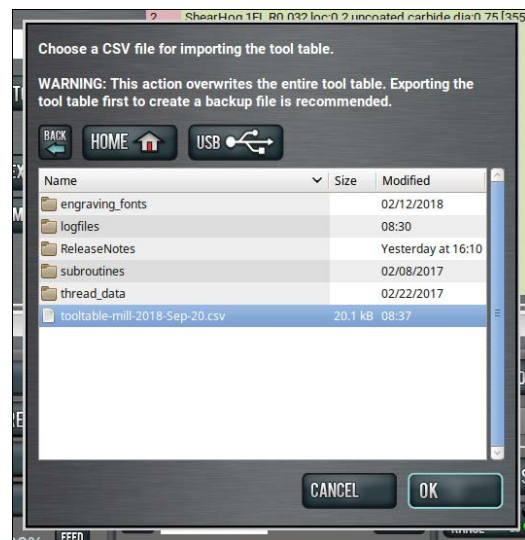


Figure 7-108: Import dialog box.

- Navigate to the .csv file on the USB drive. Then, select **OK**.

The .csv file updates the tool table.

Export the Tool Table as a .csv File

- From the **Offsets** tab, select **Export**.
PathPilot generates the .csv file, and the **Export** dialog box displays.

7: PATHPILOT TOOLS AND FEATURES

7.6 System File Management

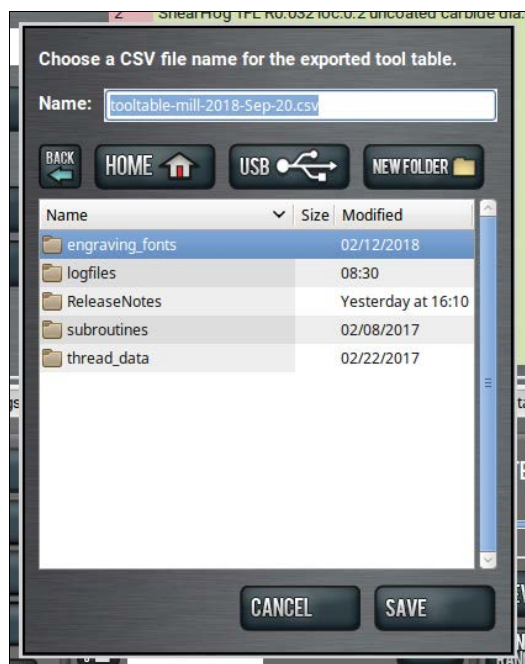


Figure 7-109: Export dialog box.

2. In the **Name** DRO field, type the name for the .csv file.
3. Select **Save**.
The .csv file is saved in the **File** tab.
4. From the **File** tab, select the newly created .csv file, and then select **Copy to USB**.
5. Select **Eject**.
It's safe to remove the USB drive from the controller.

BASIC OPERATIONS

IN THIS SECTION, YOU'LL LEARN:

- About the basic operations required for most projects, organized as a suggested project workflow.

CONTENTS

8.1 Start the Machine.....	148
8.2 Reference the Machine.....	149
8.3 Jog the Machine.....	150
8.4 Manually Control the Spindle.....	152
8.5 Load G-Code.....	153
8.6 Set Up Tooling.....	154
8.7 Set Tool Length Offsets.....	155
8.8 Set Work Offsets.....	158
8.9 Operate the Coolant Pump.....	159

8: BASIC OPERATIONS

8.1 Start the Machine

8.1 START THE MACHINE

Power on the machine and the PathPilot controller.

1. Turn the PathPilot controller strip to **ON**.
2. Turn the Main Disconnect switch to **ON** on the side of the electrical cabinet.
3. Twist out the machine's red Emergency Stop button, which enables movement to the machine axes and the spindle.
4. Press the Start button.

8.2 REFERENCE THE MACHINE

1. Verify that the machine can freely move to its reference position (at the ends of travel).
2. To verify that the tooling is clear of any possible obstructions, reference the Z-axis before referencing the other axes: from the PathPilot interface, select **Ref Z**.



Figure 8-1: Reference buttons.

3. Once the spindle is clear of any possible obstructions, continue referencing all axes.



Note: You can select the buttons one after another. Once the machine references one axis, it'll move on to the next.

After each axis is referenced, its button light comes on. For more information on referencing the machine, see "About Referencing" (page 166).

8.2.1 About Referencing

You must reference the machine to establish a known position for PathPilot. The position that's set while referencing the machine is the origin of the machine coordinate system. Without referencing the machine, PathPilot won't know the current position of the machine axes.

You must reference the machine at the following times:

- After you power on the machine
- After you push in the Emergency Stop button
- Before running a G-code program
- Before using MDI commands
- Before setting work or tool offsets
- After a collision or an axis stall/fault

When referencing, the machine moves each axis to the end of its travel. The machine stops at the limit switch, which sets the axis' reference position.

8: BASIC OPERATIONS

8.3 Jog the Machine

8.3 JOG THE MACHINE

To switch between jogging modes:

- From the **Manual Control** area, in the **Jog** group, select **Jog**.
PathPilot toggles between continuous velocity mode and step mode.



Figure 8-2: Jog button.

When the **Cont** green light is on, continuous velocity mode is selected.

When the **Step** green light is on, step mode is selected.

To use continuous velocity mode:

- Set the velocity: drag the **Jog Speed** slider.



Figure 8-3: Jog Speed slider.

For more information on continuous velocity mode, see "About Continuous Velocity Jogging" (below).

To use step mode, select the step size. Do one of the following, depending on your accessories:

- In the **Manual Control Area**, in the **Jog** group, select the step size.
The **Step** button's light comes on, indicating which step size is active.



Figure 8-4: Step buttons (in G20 mode).

- On the (optional) Jog Shuttle, press the Step button to toggle the currently selected step size.
In the PathPilot interface, the **Step** button's light comes on, indicating which step size is active.

For more information on step mode, see "About Step Jogging" (below).

8.3.1 About Jogging

Jogging is the operation of manually moving an axis in various directions (like to set up and indicate fixtures or workpieces). You can't manually jog the machine while it's performing automatic operations (like running a G-code program or an MDI command).

Jog the machine using the keyboard or the optional Jog Shuttle. Whether you're jogging with the keyboard or with the Jog Shuttle, you can either:

- Jog the machine at a consistent velocity (for information, see "About Continuous Velocity Jogging" (below)).
- Jog the machine in steps (for information, see "About Step Jogging" (below)).

For more information, see "Jog Controls Reference" (on the next page).

About Continuous Velocity Jogging

While jogging in continuous velocity mode, the machine moves at a constant speed for as long as:

- A keyboard key is pressed
- The Jog Shuttle outer ring is twisted away from the neutral position

This is useful when you're doing things like:

- Roughly positioning the machine (for example, to move the spindle head away from the workpiece).
- Moving the machine a certain distance at a constant speed (for example, to align a vise).

About Step Jogging

While jogging in step mode, the machine moves one step each time you either press a jog key on the keyboard or click the inner wheel of the Jog Shuttle. The jog step sizes range depending on the programming mode you are using:

- **Imperial (G20) Mode** 0.0001 in. to 0.1000 in.
- **Metric (G21) Mode** 0.01 mm to 10 mm

Step jogging mode is useful to finely move the machine, like when you're indicating a workpiece or manually setting tool lengths.

The jog keys on the keyboard only move the machine in steps when step mode is indicated in PathPilot. The inner wheel on the jog shuttle always moves the machine in steps, regardless of which mode is indicated in PathPilot.

8.3.2 Jog Controls Reference

The machine's jogging functions are controlled by the following:

- The Jog group of the Manual Control area in the PathPilot interface
- The keyboard
- The (optional) Jog Shuttle

Axis	Direction	Keyboard Key	Jog Shuttle
X-Axis	Positive	Right Arrow	Clockwise
	Negative	Left Arrow	Counterclockwise
Y-Axis	Positive	Up Arrow	Clockwise
	Negative	Down Arrow	Counterclockwise
Z-Axis	Positive	Page Up	Clockwise
	Negative	Page Down	Counterclockwise
A-Axis	Positive	Period	Clockwise
	Negative	Comma	Counterclockwise

Jogging in PathPilot

From the PathPilot interface, in the Manual Control area, the Jog group has the following functions:

- The Jog button, which toggles between continuous velocity mode and step mode.
- The Jog Speed slider, which controls the machine's jog rate (whether in continuous velocity mode or in step mode).
The jog rate is measured as a percentage of the machine's maximum jog rate.

Jogging with the Keyboard

Pressing the keys results in the following actions:

- **[KEY]** jogs the axis at the current jog rate.
- **[KEY]+Shift** jogs the axis at the maximum jog rate.

Jogging with the (Optional) Jog Shuttle

The optional Jog Shuttle has the following functions:

- An inner wheel and outer ring that jog the currently selected axis.
Rotating either clockwise jogs the axis in the positive direction.
Rotating either counterclockwise jogs the axis in the negative direction.

- Four axis selection buttons, which toggle the currently selected axis.



Note: The currently selected axis displays in PathPilot: in the Position Status group, there's a green light to the left of the Axis DRO field. When it's on, it indicates the active axis.

- A step button, which toggles the selected jog step size.

8: BASIC OPERATIONS

8.4 Manually Control the Spindle

8.4 MANUALLY CONTROL THE SPINDLE

1. Verify that the machine is powered on and out of reset.
2. From the PathPilot interface, in the **Manual Control** area, locate the **Spindle** group.

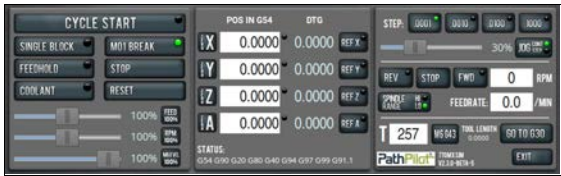


Figure 8-5: Spindle group.

3. Open the spindle motor door.
4. Make sure that the spindle belt position agrees with your desired speed range. If it doesn't, do one of the following:
 - In the PathPilot interface, select **Spindle Range**.
 - Move the spindle belt.For information, see "Change the Spindle Speed Range" (below).
5. Close the spindle motor door.
6. In the **RPM** DRO field, type the desired RPM speed. Then select the **Enter** key.
7. Select **FWD** to start the spindle in the forward direction.
8. Select **Stop** to stop the spindle.

8.4.1 Change the Spindle Speed Range

To change the spindle speed range, you must move the spindle belt inside of the spindle cabinet:

1. Open the spindle motor door.
2. Unclamp the motor mounting plate: Use a hex wrench, and pull the motor forward.
This allows the spindle motor plate to pivot.
3. Move the spindle belt from one set of pulleys to the other.
Two speed ranges are available:
 - **Low** 300 rpm to 4500 rpm
 - **High** 650 rpm to 10,000 rpm
4. Secure the motor mounting plate: Push the motor back, and tighten with a hex wrench.
5. Firmly push the spindle belt between the pulleys. If it is properly tensioned, the spindle belt should move between 3 mm and 6 mm. If it's not properly tensioned, repeat the steps in this section.

6. Close the spindle motor door.
7. From the PathPilot interface, examine the light on the **Spindle Range** button to make sure that it agrees with the spindle belt's position.



Figure 8-6: Spindle Range button.

8.4.2 About the Spindle

The machine spindle gives power to the cutting tool, which allows it to remove material from the workpiece. Operate the spindle either manually or by G-code commands (entered in the MDI Line DRO field or programmed into a G-code program). Manually controlling the spindle is useful when you're setting work offsets for tools that require the spindle to be running (like "wiggler"-style edgefinders or coaxial indicators). The machine's spindle rotates clockwise (forward) at a specified spindle speed.

8.4.3 Spindle Controls Reference

The spindle speed is measured in revolutions per minute (RPM). Two speed ranges are available:

- **Low** 300 rpm to 4500 rpm
- **High** 650 rpm to 10,000 rpm

Use the low speed range when you're using larger cutting tools, or machining materials with lower surface speeds (like steel or stainless steel). Use the high speed range when you're using smaller cutting tools, or machining materials with higher surface speeds (like aluminum, brass, or plastic). To move your spindle speed range from one to the other, see "Change the Spindle Speed Range" (above).

8.5 LOAD G-CODE

To run a G-code program on a PathPilot controller, you must first verify that the file is on the controller. For more information on transferring and moving files, see "Transfer Files to and From the Controller" (page 98).

To load G-code:

1. From the **File** tab, in the **Controller Files** window, select the desired .nc file.
2. Select **Load G-Code**.



Figure 8-7: Controller Files window on the File tab.



Note: This function is only available for files stored on the PathPilot controller.

PathPilot loads the G-code file and opens the **Main** tab.

8: BASIC OPERATIONS

8.6 Set Up Tooling

8.6 SET UP TOOLING

The Tormach Tooling System (TTS) allows you to quickly switch between various cutting tools. Before you begin machining, you must put your cutting tools in tool holders and measure the tool length offsets for each tool.

- "Install a Tool in a Set Screw Tool Holder" (page 165)
- "Install a Tool in an ER Collet Tool Holder" (page 165)
- "Install a Drill Chuck in a Jacobs Taper Arbor" (below)

8.6.1 Install a Tool in a Set Screw Tool Holder

1. Clean the shank of the tool holder with a clean rag. Verify that the shank is free of any grease or oil.
2. Remove the set screw from the tool holder with a hex wrench.
3. Put the desired cutting tool into the tool holder.
4. Replace the set screw in the tool holder, and then completely tighten it with a hex wrench.

8.6.2 Install a Tool in an ER Collet Tool Holder

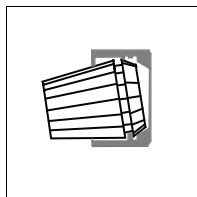
The ER20 collet is self-extracting: the collet must be mounted in the nut before the nut and collet assembly are put into the collet holder.

If you look closely, you'll notice that the collet nut isn't symmetrical — an area of the retaining ring is cut away. When the collet is correctly mounted in the nut, the collet is pushed forward and out of the collet holder taper while the nut is slightly loosened (which results in self-extraction).

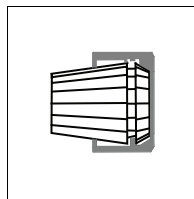
NOTICE! If you don't install the collet in the order specified, there's a risk that the collet and/or nut could be damaged, and the collet's holding capacity could be reduced.

To install a tool in an ER collet tool holder:

1. Hold the collet at an angle, and then insert it into the collet nut as shown in the following image.



2. Tilt up the collet to snap it into place.



3. Loosely thread the nut on the tool holder, insert the tool, and then tighten the collet.

8.6.3 Install a Drill Chuck in a Jacobs Taper Arbor

1. Assemble the drill chuck on to the Jacobs taper arbor:
 - a. Use a clean rag and acetone to clean the taper and socket. Verify that the taper and socket are both free of any grease or oil.
 - b. Retract the jaws: fully open the drill chuck.
 - c. Use a dead-blow hammer (or similar) to seat the drill chuck on the Jacobs taper arbor.
2. Put the drill into the drill chuck.
3. Depending on the type of drill chuck, do one of the following:
 - **Keyless Drill Chuck** Tighten the drill chuck by hand.
 - **Keyed Drill Chuck** Use a chuck key to tighten the drill chuck until it is finger tight.

8.7 SET TOOL LENGTH OFFSETS

Before running a G-code program, PathPilot must know the length of the tools that are required for the program. For more information on using tool length offsets, see "About Tool Offsets" (below).



Note: You can import a .csv file with tool length offset data. For information, see "Import and Export the Tool Table" (page 145).

To set tool length offsets:

1. Verify that the machine is powered on and out of reset.
2. Put a tool into a tool holder, and set it aside to measure. For information, see "Set Up Tooling" (page 165).
3. From the PathPilot interface, on the **Offsets** tab, verify that the **Tool** tab is selected.
4. Find the **Tool Table** window.

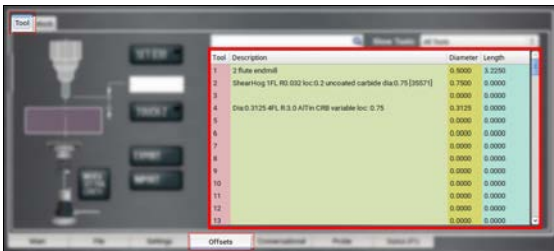


Figure 8-8: Tool Table window on the Offsets tab.

5. Depending on your workflow, you can measure your tools using any of the following methods:
 - **Use a Height Gauge** For information, see "Use a Height Gauge to Measure Tools" (page 157).
 - **Touch Off of a Known Reference Height** For information, see "Touch Off the Tool Length Offsets" (below).

8.7.1 About Tool Offsets

Tool offsets allow you to use various tools while still programming with respect to the workpiece. Tools can have different lengths (and, while using cutter radius compensation, different diameters).

The most common tool offset is the tool length offset: when you change tools, PathPilot must account for the difference in tool length. In CNC machines, the tool length offset is applied using a G43 command.

The tool length offset is the distance from the cutting edge of the tool to the shoulder of the tool holder. To make sure tool length offsets are consistent among your tools, we recommend you use Tormach Tooling System (TTS) tool holders. Set the tool length offset after placing a tool in the Tormach Tooling System (TTS) tool holder — each tool and its tool holder only need to be measured once. Before you begin a G-code program, you must verify the lengths of the tools in the program, and make sure that the lengths agree with the tool length offsets set in PathPilot:

- Each time you change tools, you must apply a new tool length offset in PathPilot.
- Each time you replace a tool, you must remeasure its length, and apply a new tool length offset in PathPilot.

NOTICE! You must always verify that the physical length of a tool agrees with the tool length offset value set in PathPilot. If you don't, there's a risk that the tool length offset misrepresents the currently active tool in the spindle, which may result in a machine crash or damaged tooling, workpieces, or fixtures.

8.7.2 Touch Off the Tool Length Offsets

Touch off the tool length offsets by using a reference surface with a known height, which gives you a basis to measure any other tool lengths. Use any surface that is parallel (within 0.02 mm) to the machine table. For example:

- A 1-2-3 Block Set (PN 31950)
- Box parallel

There are two steps to touch off the tool offsets. Complete the following steps in the order listed:

- Set a Known Reference Height 155
- Measure Tools Using a Known Reference Height 156

Set a Known Reference Height

This procedure sets a new Z zero position for the currently selected work offset.

To set a known reference height:

1. Identify a precision surface to use as a reference surface (like a 1-2-3 Block Set), and put it below the spindle on the machine table. Verify that there's a clear path from the spindle to the machine table.

8: BASIC OPERATIONS

8.7 Set Tool Length Offsets

2. Verify that the drive dogs won't contact the reference surface before the end face of the spindle.
3. Set a new, unused work offset (like G55). From the PathPilot interface, on the **Main** tab, in the **MDI Line** DRO field, type a work offset. Then select the **Enter** key. For information, see "Set Work Offsets" (page 158).
4. If there's already a tool in the spindle, remove it.
5. From the PathPilot interface, in the **Tool DRO** field, type 0. Then select the **Enter** key.
6. Slowly jog the Z-axis down (-Z) until it's 0.04 in. (1 mm) from the reference surface.
7. Measure the thickness of a piece of paper, and put the paper on the reference surface. Note the thickness of the paper for later.
8. While moving the paper back-and-forth across the reference surface, slowly step the Z-axis down (-Z) until you feel a light pull on the piece of paper. This indicates that the paper is contacting the end face of the spindle.



Note: It's easier to use step jogging for this task. For information on step jogging, see "About Step Jogging" (page 150).

9. From the PathPilot interface, in the **Z-axis** work offset DRO field, type the thickness of the piece of paper. Then select the **Enter** key.



Figure 8-9: Z-axis work offset DRO field.

The reference surface is now set as the Z zero position in the current coordinate system.

10. To set the tool length offset, go to "Measure Tools Using a Known Reference Height" (page 170).

Measure Tools Using a Known Reference Height

This procedure sets the tool length offset using a known reference height. If you have not yet done so, you must first set the Z zero position; go to Set a Known Reference Height.

To measure tools using a known reference height:

1. Verify that the reference surface is still on the machine table with the piece of paper.

2. From the PathPilot interface, on the **Offsets** tab, find an unused tool number in the **Tool Table** window. Then, type a description for the tool you're measuring.
3. Put the tool holder into the spindle.
4. From the PathPilot interface, in the **Tool DRO** field, type the number of the tool. Then select the **Enter** key.



Figure 8-10: Tool DRO field.

5. Slowly jog the Z-axis down (-Z) until it is 0.04 in. (1 mm) from the reference surface.
6. Continue to slowly jog the Z-axis while slowly moving the piece of paper back-and-forth on the reference surface.
7. Stop jogging the Z-axis when you feel a light pull on the piece of paper, which indicates that it is in contact with the tool.
8. From the PathPilot interface, on the **Offsets** tab, in the **Tool Table**, select the tool for which you previously wrote a description.
9. In the **Touch Z** DRO field, type the thickness of the piece of paper. Then select the **Enter** key.

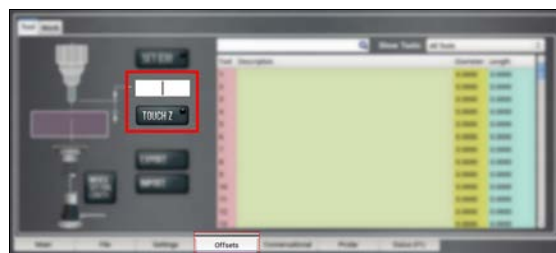


Figure 8-11: Touch Z DRO field and button.

10. Select **Touch Z**.
The length of the tool is stored in the **Tool Table** window.
11. From the **Tool Table** window, in the **Length** column, verify that the length of the tool is correct.
12. In the **Diameter** column, type the diameter of the tool. Then select the **Enter** key.

13. Jog the Z-axis up (+Z).

You've completed the procedure to measure a tool offset. Repeat this procedure for any remaining tooling you have. Once you're done adding tool length offsets, switch back to your work coordinate system.

8.7.3 Use a Height Gauge to Measure Tools

1. Before measuring a tool, you must first zero the height gauge using a precision surface, like a Granite Surface Plate (PN 31713): Move the height gauge to the top of the Granite Surface Plate.
2. Press the **Zero** button on the touch trigger.
The height gauge is zeroed.
3. Insert a tool holder into the Granite Surface Plate.
4. From the PathPilot interface, on the **Offsets** tab, select the **Tool** tab.
5. In the **Tool Table** window, type a description for the tool.
6. Move the height gauge on the Tormach Tool Assistant Set to the end of the tool's cutting edge.
7. Write down the measured tool length, and then type that value into the **Length** column on the **Tool Table** window. Then select the **Enter** key.

8: BASIC OPERATIONS

8.8 Set Work Offsets

8.8 SET WORK OFFSETS

To set the current axis location to zero in the active work coordinate system:

- Select **Zero [Axis]**.



Figure 8-12: Work Offset DRO fields.

To change work offsets:

1. On the **Main** tab, in the **MDI Line DRO** field, type the new work offset to activate (for example, G55). Then select the **Enter** key.
2. The new work offset displays in the following locations in the PathPilot interface:
 - The **Status** read-only DRO field.
 - Above the **Work Offset** DRO fields.



Figure 8-13: Work offset indicated in the PathPilot interface.

**Note:** The values in the **Work Offset** DRO fields update to indicate the new location of each axis in the new work offset.

Work Offset Naming		
Offsets 1-9 (Use either name)		
Offset	Extended Name	Name
1	G54.1 P1	G54
2	G54.1 P2	G55
3	G54.1 P3	G56
4	G54.1 P4	G57
5	G54.1 P5	G58
6	G54.1 P6	G59
7	G54.1 P7	G59.1
8	G54.1 P8	G59.2
9	G54.1 P9	G59.3
Offsets 10-500 (Use extended name)		
Offset	Extended Name	Name
10	G54.1 P10	Not used
11	G54.1 P11	Not used
...		
499	G54.1 P499	Not used
500	G54.1 P500	Not used

For more information on using work offsets, see "About Work Offsets" (page 168).

8.8.1 About Work Offsets

Work offsets allow you to think in terms of X, Y, and Z coordinates with respect to the part, rather than thinking of them with respect to the machine position. This means that you can jog the machine to an arbitrary location (like the end of a workpiece) and call that location zero. You can save up to 500 work offsets in PathPilot. The naming structure varies based on the offset number, as detailed in the following table.

8.9 OPERATE THE COOLANT PUMP

To turn coolant on or off:

- Select **Coolant**.



Figure 8-14: Coolant button.

For more information on turning on and off coolant, see "About Coolant" (below).

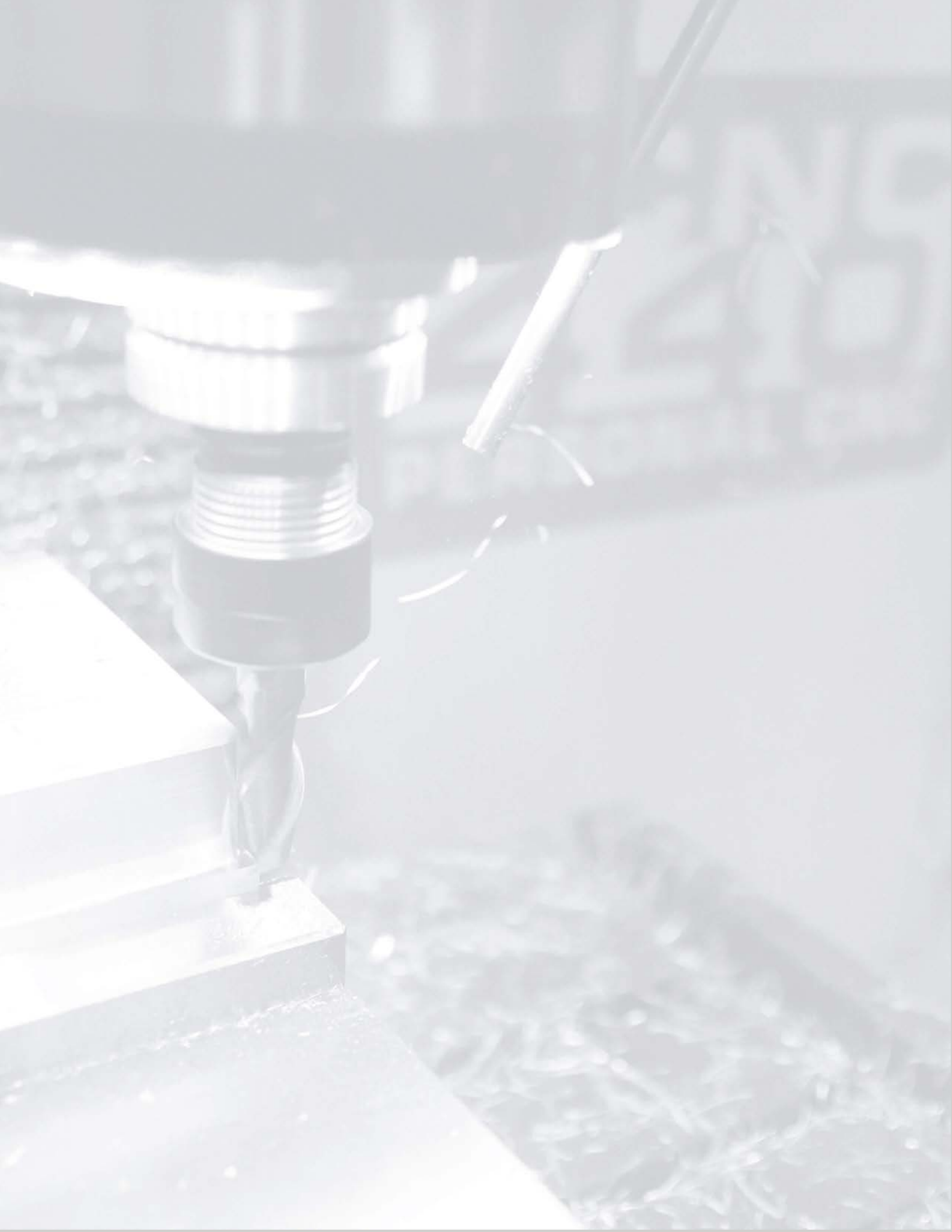
8.9.1 About Coolant

In the PathPilot interface, the Coolant button controls the machine's coolant pump power outlet. The Coolant button's light shows the current state of the outlet: the light is on when the outlet has power; the light is off when the outlet does not have power.



Note: The Coolant button is equivalent to using an M08 (coolant on) or M09 (coolant off) command in the G-code program.

Use the Coolant button before or after a program is running, while a program is running, or while you are using manual data input (MDI) commands.



FIRST PART TUTORIAL

IN THIS SECTION, YOU'LL LEARN:

- How to complete your first project, with all specifications provided by us.

CONTENTS

9.1 About This Tutorial	162
9.2 Before You Begin	163
9.3 Required Items and Tools	164
9.4 Set Up Tooling	165
9.5 Set Up the Machine	166
9.6 Mount the Workpiece	167
9.7 Set the Work Offsets	168
9.8 Set Tool Offsets	170
9.9 Complete the Milling Operations	171

9: FIRST PART TUTORIAL

9.1 About This Tutorial

9.1 ABOUT THIS TUTORIAL

This tutorial project is a shallow circular pocket with the letters PCNC engraved into a 2 in. × 4 in. piece of wood.



Figure 9-1: Example of a completed first part tutorial.



Tip! Using a piece of wood reduces the possibility of damaging a tool if you enter an incorrect work or tool offset.

9.2 BEFORE YOU BEGIN

This tutorial assumes that you have no prior experience running a part program on a computer numerically controlled (CNC) machine. This tutorial will give you an introduction to the controls of the machine using PathPilot.

The two tools used give an introduction to tool changes and the difference between work offsets and tool length offsets. If you have an (optional) Automatic Tool Changer (ATC), we recommend using manual tool changes in this tutorial to keep things simple.

9: FIRST PART TUTORIAL

9.3 Required Items and Tools

9.3 REQUIRED ITEMS AND TOOLS

Before you begin, collect the following items and tools:

- A scrap piece of wood (at least 2 in. × 4 in. × 4 in.)
- 1/8 in. end mill
- 3/8 in. end mill



Tip! The Two Flute High-Speed Steel End Mill

Set (PN 33465) includes both end mills, and three extra — it's a good set to start with.

- Marker
- TTS Set Screw Holder: 3/8 in. (PN 31820)

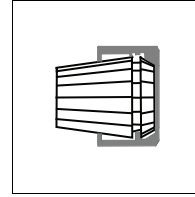


Tip! You can also use an ER16 or ER20 collet chuck.

9.4 SET UP TOOLING

The Tormach Tooling System (TTS) allows you to quickly switch between various cutting tools. Before you begin machining, you must put your cutting tools in tool holders and measure the tool length offsets for each tool.

- "Install a Tool in a Set Screw Tool Holder" (below)
- "Install a Tool in an ER Collet Tool Holder" (below)
- "Install a Drill Chuck in a Jacobs Taper Arbor" (page 154)



3. Loosely thread the nut on the tool holder, insert the tool, and then tighten the collet.

9.4.1 Install a Tool in a Set Screw Tool Holder

1. Clean the shank of the tool holder with a clean rag. Verify that the shank is free of any grease or oil.
2. Remove the set screw from the tool holder with a hex wrench.
3. Put the desired cutting tool into the tool holder.
4. Replace the set screw in the tool holder, and then completely tighten it with a hex wrench.

9.4.2 Install a Tool in an ER Collet Tool Holder

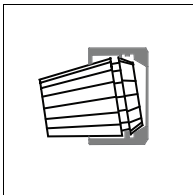
The ER20 collet is self-extracting: the collet must be mounted in the nut before the nut and collet assembly are put into the collet holder.

If you look closely, you'll notice that the collet nut isn't symmetrical — an area of the retaining ring is cut away. When the collet is correctly mounted in the nut, the collet is pushed forward and out of the collet holder taper while the nut is slightly loosened (which results in self-extraction).

NOTICE! If you don't install the collet in the order specified, there's a risk that the collet and/or nut could be damaged, and the collet's holding capacity could be reduced.

To install a tool in an ER collet tool holder:

1. Hold the collet at an angle, and then insert it into the collet nut as shown in the following image.



2. Tilt up the collet to snap it into place.

9: FIRST PART TUTORIAL

9.5 Set Up the Machine

9.5 SET UP THE MACHINE

1. Power on the machine and the PathPilot controller.
 - a. Turn the Main Disconnect switch to **ON** on the side of the electrical cabinet.
 - b. Twist out the machine's red Emergency Stop button, which enables movement to the machine axes and the spindle.
 - c. Press the button.
2. Verify that the machine can freely move to its reference position (at the ends of travel).
3. To verify that the tooling is clear of any possible obstructions, reference the Z-axis before referencing the other axes: from the PathPilot interface, select **Ref Z**.



Figure 9-2: Reference buttons.

4. Once the spindle is clear of any possible obstructions, continue referencing all axes.

Note: You can select the buttons one after another. Once the machine references one axis, it'll move on to the next.

After each axis is referenced, its button light comes on. For more information, go to "About Referencing" (below).

9.5.1 About Referencing

You must reference the machine to establish a known position for PathPilot. The position that's set while referencing the machine is the origin of the machine coordinate system. Without referencing the machine, PathPilot won't know the current position of the machine axes.

You must reference the machine at the following times:

- After you power on the machine
- After you push in the Emergency Stop button
- Before running a G-code program
- Before using MDI commands
- Before setting work or tool offsets
- After a collision or an axis stall/fault

When referencing, the machine moves each axis to the end of its travel. The machine stops at the limit switch, which sets the axis' reference position.

9.6 MOUNT THE WORKPIECE

- Put a piece of scrap wood (2 in. × 4 in.) in the vise. Make sure that the top of the workpiece is at least 1/4 in. above the top of the vise jaws.

9: FIRST PART TUTORIAL
9.7 Set the Work Offsets

9.7 SET THE WORK OFFSETS

Touch off of the workpiece's center to establish the Z work offsets. Then, use a tool to help you establish the X and Y work offsets.

9.7.1 Set the Z Zero

This procedure sets a new Z zero position for the currently selected work offset.

To set a known reference height:

- 1. If there's already a tool in the spindle, remove it.
- 2. Use a marker to mark the approximate center of the workpiece.
- 3. From the PathPilot interface, in the Tool DRO field, type 0. Then select the Enter key.

Tip! It's important that there's no tool in the spindle, and that there's no tool commanded in PathPilot. If a tool was loaded into the spindle, or even commanded in PathPilot, the Z offset would account for the programmed tool length.

- 4. Slowly jog the Z-axis down (-Z) until it's 0.04 in. (1 mm) from the workpiece.
- 5. Put a piece of paper on the workpiece.
- 6. While moving the paper back-and-forth across the workpiece, slowly step the Z-axis down (-Z) until you feel a light pull on the piece of paper. This indicates that the paper is contacting the spindle.

Note: It's easier to use step jogging for this task. For information on step jogging, see "About Step Jogging" (page 150).



Figure 9-3: Example of using a piece of paper on the workpiece.

- 7. From the PathPilot interface, in the Z-axis work offset DRO field, type the thickness of the piece of paper. Then select the Enter key.



Figure 9-4: Z-axis work offset DRO field.

The reference surface is now set as the Z zero position in the current coordinate system.

9.7.2 Set the X/Y Zeroes

- 1. Put tool 1 (the 3/8-in. end mill) into the spindle.
- 2. Jog the tool so that it's close to the mark you put on the workpiece.
- 3. From the PathPilot interface, select Zero X. Then, select Zero Y.



Figure 9-5: Zero X and Zero Y buttons.

9.7.3 About Work Offsets

Work offsets allow you to think in terms of X, Y, and Z coordinates with respect to the part, rather than thinking of them with respect to the machine position. This means that you can jog the machine to an arbitrary location (like the end of a workpiece) and call that location zero. You can save up to 500 work offsets in PathPilot. The naming structure varies based on the offset number, as detailed in the following table.

Work Offset Naming		
Offsets 1-9 (Use either name)		
Offset	Extended Name	Name
1	G54.1 P1	G54
2	G54.1 P2	G55
3	G54.1 P3	G56

Work Offset Naming		
4	G54.1 P4	G57
5	G54.1 P5	G58
6	G54.1 P6	G59
7	G54.1 P7	G59.1
8	G54.1 P8	G59.2
9	G54.1 P9	G59.3
Offsets 10-500 (Use extended name)		
Offset	Extended Name	Name
10	G54.1 P10	Not used
11	G54.1 P11	Not used
...		
499	G54.1 P499	Not used
500	G54.1 P500	Not used

9.7.4 About True Positive Tool Length

To conceptualize tool and work offsets, we use the idea of a true positive tool length. This means that, if you use the face of the empty spindle to set your work offset and touch off your tools, the tool length offsets will be equal in value to the length of the tool.

There are a few benefits to using true positive tool lengths when measuring tool offsets:

- Looking at the tool length value, you can estimate if it's correct for a given tool by checking with a ruler or calipers.
- All of your tools don't have to be measured the same way — you can mix tools touched off on the machine with tools measured using a digital height gauge.
- Conceptually, it's easier to understand.

9.7.5 About X/Y Part Zeroes

Common positions for the X and Y part zeros are:

- The back left of the workpiece
- The center of the workpiece
- A feature, like a hole or a boss, that already exists on the workpiece

9: FIRST PART TUTORIAL

9.8 Set Tool Offsets

9.8 SET TOOL OFFSETS

9.8.1 Touch Off the Tool Offsets

1. Make sure that tool 1 (the 3/8 in. end mill) is in the spindle.
2. In the **Tool DRO** field, type 1. Then, select **M6 G43**. This tells the machine that you've changed tools, and want to apply the tool length offset.
3. Jog the tool so that it's touching the workpiece.
4. From the PathPilot interface, on the Offsets tab, type 0 in the **Touch DRO** field. Then, select **Touch Z**.



Tip! If you weren't touching on the top of the workpiece, but instead using a piece of paper between the workpiece and the tool, you would enter the thickness of the paper in the **Touch DRO** field before you selected **Touch Z** to account for the paper thickness.

5. In the **Tool Table** window, identify the length entered for tool 1.
6. Use a calipers to verify that the length for tool 1 is correct.
7. In the **Tool Table** window, type the diameter of the tool. Then select the **Enter** key.



Note: Fractions are converted to their decimal equivalents.

8. Put tool 2 (the 1/8 in. end mill) into the spindle.
9. Repeat Steps 2 through 7 to measure the tool length for tool 2.

9.8.2 Measure Tools Using a Known Reference Height

This procedure sets the tool length offset using a known reference height. If you have not yet done so, you must first set the Z zero position; go to Set a Known Reference Height. To measure tools using a known reference height:

1. From the PathPilot interface, on the **Offsets** tab, find an unused tool number in the **Tool Table** window. Then, type a description for the tool you're measuring.
2. Put the tool holder into the spindle.

3. From the PathPilot interface, in the **Tool DRO** field, type the number of the tool. Then select the **Enter** key.



4. From the PathPilot interface, on the **Offsets** tab, in the **Tool Table**, select the tool for which you previously wrote a description.
5. Select **Touch Z**.
The length of the tool is stored in the **Tool Table** window.
6. From the **Tool Table** window, in the **Length** column, verify that the length of the tool is correct.
7. In the **Diameter** column, type the diameter of the tool. Then select the **Enter** key.

9.9 COMPLETE THE MILLING OPERATIONS

There are two milling operations required for this tutorial.
Complete the following steps in the order listed:

- 9.9.1 Create a Pocket 171
- 9.9.2 Engrave the Text 172

9.9.1 Create a Pocket

The first step is to mill a pocket into the face of the workpiece that is 0.100 in. deep and 3-1/4 in. in diameter.

Load the Tool

- Put tool 1 (the 3/8 in. end mill) into the spindle.
- From the PathPilot interface, in the **Tool DRO** field, type 1. Then select the **Enter** key.



Figure 9-6: Tool DRO field.

Write the G-Code with Conversational

- From the **Conversational** tab, select the **Pocket** tab.
- On the **Pocket** tab, select **Rect/Circ**.
PathPilot switches the interface to create a circular pocket.
- Type the following values into the DRO fields, and press **Enter** on the keyboard after typing each value:
 - In the **Title** DRO field, type **FIRST PART POCKET**.
 - In the **Work Offset** DRO field, type **G54**.
 - In the **Tool** DRO field, type **1**.
 - In the **Spindle** DRO field, type **5000**.
 - In the **Feedrate** DRO field, type **60**.
 - In the **Z Feedrate** DRO field, type **30**.
 - In the **Z Clear** DRO field, type **0.1000**.
 - In the **Pocket Dia.** DRO field, type **3.2500**.
 - In the **Z Start** DRO field, type **0.000**.
 - In the **Z End** DRO field, type **-0.1000**.
 - In the **Z DOC** DRO field, type **0.1000**.
 - In the **Stepover** DRO field, type **0.2500**.

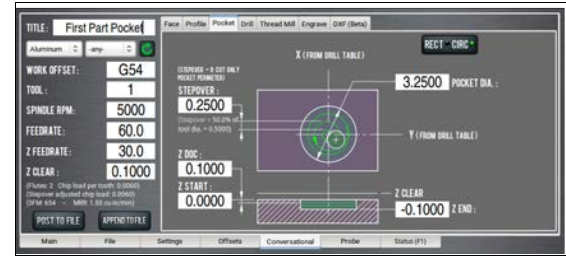


Figure 9-7: Values typed into the DRO fields on the Pocket tab of the Conversational tab.

- From the **Conversational** tab, select the **Drill/Tap** tab.
- On the **Drill/Tap** tab, make sure that the **Pattern** tab is selected. Then, type the following values into the following DRO fields:
 - In the DRO field in the **X** column of row **1**, type **0.0000**. Then select the **Enter** key.
 - In the DRO field in the **Y** column of row **1**, type **0.0000**. Then select the **Enter** key.

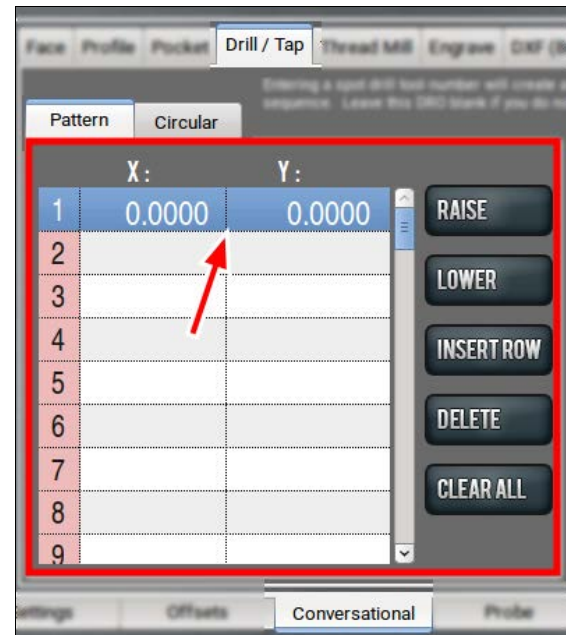


Figure 9-8: Values typed into the DRO fields on the Pattern tab of the Drill/Tap tab.

- From the **Conversational** tab, select the **Pocket** tab.

9: FIRST PART TUTORIAL

9.9 Complete the Milling Operations

7. Select **Post to File**.

PathPilot generates the G-code file, and the **Save** dialog box displays.



Figure 9-9: Save dialog box.

8. From the **Save** dialog box, specify where on the PathPilot controller to save the G-code file.

9. Select **Save**.

PathPilot loads the G-code file and opens the **Main** tab.

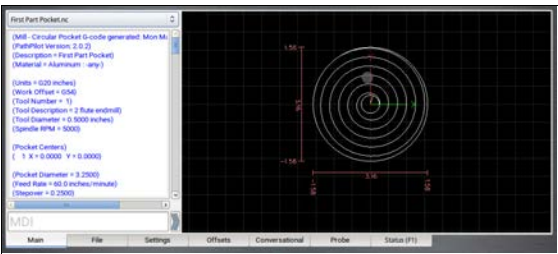


Figure 9-10: Example of the pocket program.

Run the Program

1. Restrict the speed of the mill so that you can more easily watch the operation running: Set the **Maxvel Override** slider to **0%**.

Tip! If you have an optional ATC, this step stops all motion, and prevents the machine from changing tools.

2. Select **Cycle Start**.

The spindle begins to rotate, but the tool won't move, because you restricted the speed in Step 1.

3. Change the value of the **Maxvel Override** slider to speed up, slow down, or stop the mill. When the tool is close to the part, set the **Maxvel Override** slider to **0%**.

4. On the **Main** tab, verify the values in the DRO fields to make sure that they look correct for the current tool

position. Then, slowly move the **Maxvel Override** slider to speed up the mill.

EXAMPLE

If the value in the **Z DRO** field is **0.2500**, the tool should be 1/4 in. above the workpiece.

5. After the operation is complete, examine the top surface of the workpiece to make sure that the entire pocket has been cut by the end mill.

9.9.2 Engrave the Text

Write the G-Code with Conversational

1. From the PathPilot interface, on the **Engrave** tab, type the following values into the DRO fields, and press **Enter** on the keyboard after typing each value:
- a. In the **Title** DRO field, type **FIRST PART POCKET**.
 - b. In the **Work Offset** DRO field, type **G54**.
 - c. In the **Tool** DRO field, type **2**.
 - d. In the **Spindle RPM** DRO field, type **5000**.
 - e. In the **Feedrate** DRO field, type **60**.
 - f. In the **Z Clear** DRO field, type **0.1000**.
 - g. In the **Text** DRO field, type **PCNC**.
 - h. In the **Height** DRO field, type **0.7500**.
 - i. In the **X Base** DRO field, type **0.0000**.
 - j. In the **Y Base** DRO field, type **-0.3750**.
 - k. In the **Z Start** DRO field, type **-0.1000**.
 - l. In the **Z Depth of Cut** DRO field, type **0.0500**.



Figure 9-11: Example of the values to type on the Engrave tab of the Conversational tab.

2. From the **Font** list, select **FreeMonoOblique.ttf**.
3. From the **Alignment** radio buttons, select **Center**.

4. Select **Append to File**.

PathPilot generates the G-code file, and the **Append to File** dialog box displays.



Figure 9-12: Append to File dialog box.

5. From the **Append to File** dialog box, specify to which file you want to add this operation.
6. Select **Append to File**.
PathPilot adds the G-code to the file and opens the **Main** tab.

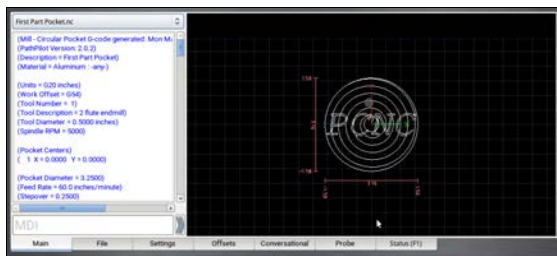


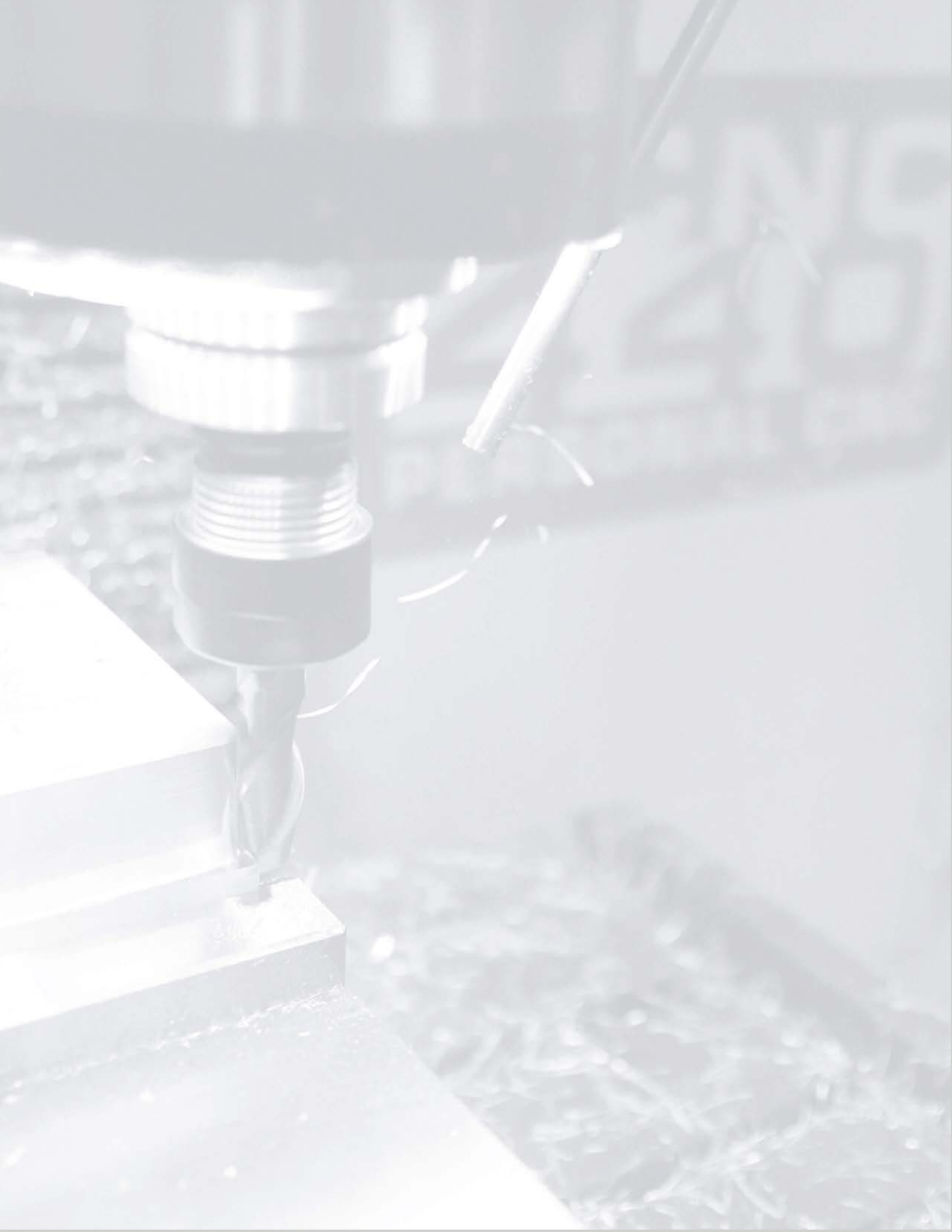
Figure 9-13: Example of the engrave and pocket program.

Run the Program

Tip! Because you appended the second operation onto the first, the program starts from the beginning — it re-cuts the pocket. To avoid this in the future, you can post the second operation to a separate file.

1. Select **Cycle Start**.
The spindle begins to rotate, and the tool moves to the beginning of the tool path.
2. After the operation is complete, put tool 2 (the 1/8 in. end mill) into the spindle.
3. From the PathPilot interface, in the **Tool DRO** field, type 2. Then select the **Enter** key.

4. Restrict the speed of the mill so that you can more easily watch the operation running: Set the **Maxvel Override** slider to **0%**.
5. Select **Cycle Start**.
The spindle begins to rotate, and the tool moves to the beginning of the tool path.
6. Change the value of the **Maxvel Override** slider to speed up, slow down, or stop the speed of the mill.
7. After the operation is complete, examine the top surface of the workpiece to make sure that the entire phrase has been engraved by the end mill.



PROGRAMMING

IN THIS SECTION, YOU'LL LEARN:

- About the languages that are understood and interpreted by PathPilot.

CONTENTS

10.1 Programming Overview.....	176
10.2 Programming G-Code.....	182
10.3 Programming Canned Cycles.....	194
10.4 Programming M-Code.....	199
10.5 Programming Input Codes.....	203
10.6 Advanced Programming.....	204

10: PROGRAMMING

10.1 Programming Overview

10.1 PROGRAMMING OVERVIEW

Read the following sections for a G-code overview:

10.1.1 About G-Code Programming Language	176
10.1.2 G-Code Formatting Reference	176
10.1.3 Supported G-Codes Reference	180

10.1.1 About G-Code Programming Language

A G-code program is made up of one or more lines of code. Each line of code is called a block, and can include commands to the machine. Blocks are collected into a file, which makes a program.

A block is normally made up of an optional line number at the beginning, followed by one or more words, which groups the elements together into a single statement.

A word is a letter followed by a number (or, something that evaluates to a number). A word can either give a command or provide an argument to a command.

A program is one or more blocks, each separated by a line break. Blocks in a program are executed either:

- Sequentially (from the top of the program to the bottom)
- Until an end command (M02 or M30) is encountered

EXAMPLE :

G01 X3 is a valid line of code with two words:

- **G01** is a command: the machine should move in a straight line at the programmed feed rate.
- **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) — G-codes and M-codes.

There are two commands (M02 and M30) that end a program. A program can end before the end of a file. If there are lines in a file after the end of a program, they're not meant to be executed in the normal flow (they're generally parts of subroutines).

10.1.2 G-Code Formatting Reference

A permissible block of input code is made up of the following programming elements, in order, with the restriction that there is a maximum of 256 characters allowed on a line:

1. (Optional) Block delete character (/)
2. (Optional) Line number

3. Any number of words, parameter settings, and comments
4. End of line marker (carriage return or line break)

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. Blank lines are allowed in the input, but they're ignored. Input is not case sensitive (except in comments), so any letter outside a comment may be in uppercase or lowercase without changing the meaning of a line.



EXAMPLE

```
G00 x +0. 12 34y 7 is equal to G00 x+0.1234
y7
```

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.
- Zero to four M words, but 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.

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

PathPilot omits blocks of code that are prefixed with a block delete character (/).

PathPilot sometimes ignores things it doesn't understand. If a command doesn't work as expected, or does nothing, make sure that it's correctly typed. PathPilot doesn't check for excessively high machining feeds or speeds, and it doesn't detect situations where a legal command will do something unfortunate (like machining a fixture).

Line Numbers

A line number is indicated by the following, in the order listed:

1. The letter N
2. An integer (with no sign) between 0 and 99,999,999 (which must be written without commas)

Line numbers may be repeated, or used out of order, but that's rare in normal practice. A line number isn't required, and is often omitted.

Words

A word is indicated by the following, in the order listed:

1. A letter other than N or O
2. A real value

Letters

Words may begin with any of the following letters, except N or O:



Note: Several letters (I, J, K, L, P and R) may have different meanings in different contexts.

Letter	Description
A	A-axis
B	B-axis
C	C-axis
D	Tool radius compensation number
F	Feed rate
G	General function
H	Tool length offset index
I	X-axis offset for arcs
J	Y-axis offset for arcs
K	Z-axis offset for arcs
L	Number of repetitions in canned cycles and subroutines, or key used with G10
M	Miscellaneous function
N	Line number
O	Subroutine label number
P	Dwell time in canned cycles, dwell time with G04, key used with G10, or tapping depth in M871 through M874
Q	Feed increment in a G83 canned cycle, or repetitions of subroutine call
R	Arc radius, or canned cycle retract level
S	Spindle speed
T	Tool selection
U	Synonymous with A

Letter	Description
V	Synonymous with B
W	Synonymous with C
X	X-axis
Y	Y-axis
Z	Z-axis

Values

A real value is one of the following:

- An explicit number (like 341, or -0.8807)
- An expression (like [2+2.4])
- A parameter value (like #88)
- A unary operation value (like acos[0])



Note: In the command examples that we use, the tilde symbol (~) stands for a real value. If $L\sim$ is written in an example, the ~ is often referred to as the L number. Similarly the ~ in $H\sim$ may be called the H number, and so on for any other letter.

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

Explicit numbers have the following rules (in this case, a digit is a single character, 0 through 9):

- A number must consist of the following, in the order listed:
 1. An optional plus or minus sign
 2. Zero to many digits
 3. (Optional) One decimal point
 4. Zero to many digits
- There must be at least one digit somewhere in the number.
- It must be either an integer (no decimal point) or a decimals (decimal point).

10: PROGRAMMING

10.1 Programming Overview

- It may have any number of digits (subject to line length limitations).



Note: PathPilot only keeps 17 significant figures, which is enough for all known applications.

- A non-zero number with no sign as the first character is assumed to be positive.

Initial zeros (a zero before the decimal point and the first non-zero digit) and trailing zeros (a zero 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 PathPilot are often restricted to some finite set of values, or to some range of values. In many uses, decimal numbers must be close enough to an integer to be accepted as a valid 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.

Order of Execution

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

The order of items on a line doesn't determine the order of execution on the commands.

Three types of items' order may vary on a line (as given earlier in this section):

- Word** May be reordered in any way without changing the meaning of the line.
- Parameter Setting** If it's 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.



EXAMPLE

When the line #3=15 #3=6 is 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.

- Comment** If it 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.



EXAMPLE

G40 G01 #3=15 (f00) #4=-7.0 has five items and means exactly the same thing in any of the 120 possible orders, like #4=-7.0 G01 #3=15 G40 (f00), for the five items.

The order of execution of items on a line is critical to safe and effective machine operation. If items occur on the same line, they are executed in a particular order. To impose a different order (like to turn coolant off before the spindle is stopped), code the commands on separate blocks.

The order of execution is as follows:

- Comment (including message)
- Set feed rate mode (G93, G94, G95)
- Set feed rate (F)
- Set spindle speed (S)
- Special I/O (M62 to M68)



Note: This is not supported.

- Change tool (T)
- Spindle on/off (M03, M05)
- Save State (M70, M73, restore state (M72), invalidate state (M71)
- Coolant on/off (M07, M08, M09)
- Enable/disable overrides (M48, M49, M50, M51, M52, M53)
- Operator defined commands (M101 to M199)
- Dwell (G04)
- Set active plane (G17, G18, G19)
- Set length units (G20, G21)
- Cutter radius compensation on/off (G40, G41, G42)
- Tool table offset on/off (G43, G49)
- Fixture table select (G54 through G58 and G59 P~)
- Set path control mode (G61, G61.1, G64)
- Set distance mode (G90, G91)
- 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)

Modal Groups

G- and M-codes are, generally speaking, modal — they cause the machining system to change from one mode to another. The mode stays active until another command changes it implicitly or explicitly.



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 (like Dwell (G04)). These codes have effect only on the lines on which they occur. Modal commands are arranged in sets, called modal groups. 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 (like 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. 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.

Modal groups for G-codes are detailed in the following table.

Group	Commands	Group Description
Group 1	{G00, G01, G02, G03, G33, G38.x, G73, G76, G80, G81, G82, G84, G85, G86, G88, G89}	Motion (one always in effect)

Group	Commands	Group Description
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 distance mode
Group 5	{G93, G94}	Feed rate mode
Group 6	{G20, G21}	Length units
Group 7	{G40, G41, G42, G41.1, G42.1}	Cutter 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}	Select work offset coordinate system
Group 13	{G61, G61.1, G64}	Path control mode
Group 14	{G96, G97}	Spindle control mode
Group 15	{G07, G08}	Lathe diameter mode

Modal groups for M-codes are detailed in the following table.

Group	Commands	Group Description
Group 4	{M00, M01, M02, M30, M60}	Program stop and program end
Group 7	{M03, M05}	Spindle control

10: PROGRAMMING

10.1 Programming Overview

Group	Commands	Group Description
Group 8	{M07, M08, M09}	Coolant control (special case: M07 and M08 may be active at the same time)
Group 9	{M48, M49}	Override control

Non-modal G-codes are:

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

Comments

You can add comments to lines of G-code to help clarify the intention of the programmer. To embed a comment in a line, use parentheses. To add a comment to the end of a line, use a semicolon.



Note: The semicolon is not treated as the start of a comment when it's enclosed in parentheses.

Comments can appear between words, but they can't be between words and their corresponding parameter.



EXAMPLE :

S100 (set speed) F200 (feed) is okay, but S (speed) 100F (feed) is not.

10.1.3 Supported G-Codes Reference

G-Code	Description
G00	Rapid linear motion
G01	Linear motion at feed rate
G02	Clockwise arc at feed rate
G03	Counterclockwise arc at feed rate
G04	Dwell

G-Code	Description
G07, G08	Diameter / radius mode Note: The 15L Slant-PRO lathe and the RapidTurn both use G07 (X positions displayed in diameter values). G08 is not used or supported in PathPilot.
G10 L1	Set tool table
G10 L2	Set coordinate system
G10 L10	Set tool table – calculated – workpiece
G10 L11	Set tool table – calculated – fixture
G10 L20	Set coordinate system
G17, G18, G19	Plane selection
G20/G21	Length units
G28	Return to predefined position
G28.1	Return to predefined position
G30	Return to predefined position
G33	Spindle synchronized motion (like threading)
G40	Cancel cutter compensation
G41/G42	Cutter compensation (left/right)
G41.1, G42.1	Dynamic cutter compensation
G43	Apply tool length offset
G49	Cancel tool length compensation
G53	Absolute coordinates
G54-G59.3	Select work offset coordinate system
G61/G61.1	Set exact path control mode
G64	Set blended path control mode
G73	High-speed peck drill
G76	Multi-pass threading cycle
G80	Cancel canned cycles

G-Code	Description
<u>G81</u>	Drilling cycle
<u>G82</u>	Simple drilling cycle
<u>G83</u>	Peck drilling cycle
<u>G85</u>	Boring cycle
<u>G86</u>	Boring cycle
<u>G88</u>	Boring cycle
<u>G89</u>	Boring cycle
<u>G90,</u> <u>G90.1</u>	Arc distance mode
<u>G91,</u> <u>G91.1</u>	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92, etc.
<u>G93, G94,</u> <u>G95</u>	Feed rate mode
<u>G96, G97</u>	Spindle control mode
G98	Initial level return / R-point level after canned cycles

10: PROGRAMMING

10.2 Programming G-Code

10.2 PROGRAMMING G-CODE

Read the following sections as a G-code reference:

10.2.1 About the Examples Used	182
10.2.2 Rapid Linear Motion (G00).....	182
10.2.3 Linear Motion at Feed Rate (G01).....	182
10.2.4 Arc at Feed Rate (G02 and G03).....	183
10.2.5 Dwell (G04).....	185
10.2.6 Set Offsets (G10).....	185
10.2.7 Plane Selection (G17, G18, G19).....	186
10.2.8 Length Units (G20 and G21).....	186
10.2.9 Return to Predefined Position (G28 and G28.1) ...	186
10.2.10 Return to Predefined Position (G30 and G30.1) ...	187
10.2.11 Automatically Measure Tool Lengths with an ETS (G37 and G37.1).....	187
10.2.12 Straight Probe (G38.x).....	188
10.2.13 Cutter Compensation (G40, G41, G42).....	189
10.2.14 Dynamic Cutter Compensation (G41.1 and G42.1).....	189
10.2.15 Apply Tool Length Offset (G43).....	189
10.2.16 Engrave Sequential Serial Number (G47).....	190
10.2.17 Cancel Tool Length Compensation (G49).....	190
10.2.18 Absolute Coordinates (G53).....	190
10.2.19 Select Work Offset Coordinate System (G54 to G54.1 P500).....	190
10.2.20 Set Exact Path Control Mode (G61).....	191
10.2.21 Set Blended Path Control Mode (G64).....	191
10.2.22 Distance Mode (G90 and G91).....	191
10.2.23 Arc Distance Mode (G90.1 and G91.1).....	191
10.2.24 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	191
10.2.25 Feed Rate Mode (G93, G94, and G95).....	192
10.2.26 Spindle Control Mode (G96 and G97).....	192

10.2.1 About the Examples Used

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

- 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.

10.2.2 Rapid Linear Motion (G00)

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

- X~ is the X-axis coordinate
- Y~ is the Y-axis coordinate
- Z~ is the Z-axis coordinate
- A~ is the A-axis coordinate

This produces coordinated linear motion to the destination point at the current traverse rate (or slower, if the machine won't go that fast). It's expected that cutting won't take place when a G00 command is executing. The G00 is optional if the current motion mode is G00.

Depending on where the tool is located, follow these two basic rules:

1. If the Z value represents a cutting move in the positive direction (like out of a hole), the X-axis should be moved last.
2. If the Z value represents a move in the negative direction, the X-axis should be moved first.

Conditions

The motion differs if:

- Cutter radius compensation is active
- G53 is programmed on the same line

Troubleshooting

It's an error if:

- All axis words are omitted
The axis words are optional, except that at least one must be used.
- G10, G28, G30 or G92 appear in the same block

10.2.3 Linear Motion at Feed Rate (G01)

For linear motion at feed rate (for cutting or not), program:

`G01 X~ Y~ Z~ A~ F~`

- X~ is the X-axis coordinate
- Y~ is the Y-axis coordinate
- Z~ is the Z-axis coordinate
- A~ is the A-axis coordinate
- F~ is the feed rate

This produces coordinated linear motion to the destination point at the current feed rate (or slower, if the machine won't go that fast). The G01 is optional if the current motion mode is G01.

Conditions

The motion differs if:

- Cutter radius compensation is active
- G53 is programmed on the same line

Troubleshooting

It's an error if:

- All axis words are omitted
The axis words are optional, except that at least one must be used.
- G10, G28, G30, or G92 appear in the same block
- No F word is specified

10.2.4 Arc at Feed Rate (G02 and G03)

A circular or helical arc is specified using either G02 (clockwise arc) or G03 (counterclockwise arc). The axis of the circle or helix must be parallel to the X-, Y- or Z-axis of the machine 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. This is rare.

The motion differs if cutter radius compensation is active.

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's the current motion mode.

Radius Format Arc

For a clockwise arc in radius format, program: G02 X~ Y~ Z~ A~ R~

For a counterclockwise arc in radius format, program: G03 X~ Y~ Z~ A~ R~

- X~ is the X-axis coordinate
- Y~ is the Y-axis coordinate
- Z~ is the Z-axis coordinate
- A~ is the A-axis coordinate
- R~ is the radius of the arc

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. 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. We don't recommend programming radius format arcs that are:

- Nearly full circles
- Semicircles
- Nearly semicircles

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 the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts.

You can program arcs that are:

- Up to 165 degrees
- Between 195 degrees to 345 degrees



EXAMPLE

G17 G02 X 1.0 Y 1.5 R 2.0 Z 0.5 is a radius format command to mill an arc, which makes 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's a helical arc.

Troubleshooting

It's an error if:

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

Center Format Arc

For a clockwise arc in center format, program: G02 X~ Y~ Z~ I~ J~

For a counterclockwise arc in center format, program: G03 X~ Y~ Z~ I~ J~

- X~ is the X-axis coordinate
- Y~ is the Y-axis coordinate
- Z~ is the Z-axis coordinate

10: PROGRAMMING

10.2 Programming G-Code

- **A~** is the A-axis coordinate
- **I~** is the center of arc (X coordinate)
- **J~** is the center of arc (Y coordinate)
- **K~** is the center of arc (Z 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's okay if the end point of the arc is the same as the current point.

The center is specified using the 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's 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 you're programming in inches) or 0.002 millimeters (if you're programming in millimeters)
- **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**)

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).

It's an error if:

- X and Y are both omitted
The axis words are all optional except that at least one of X and Y must be used.
- I and J are both omitted
I and J are optional except that at least one of the two must be used.

Arc in XZ Plane

When the XZ-plane is selected, program: **G02 X~ Y~ Z~ A~ I~ K~** (or, use **G03** instead of **G02**)

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).

It's an error if:

- X and Z are both omitted
The axis words are all optional except that at least one of X and Z must be used.
- I and K are both omitted
I and K are optional except that at least one of the two must be used.

Arc in YZ Plane

When the YZ-plane is selected, program: **G02 X~ Y~ Z~ A~ J~ K~** (or, use **G03** instead of **G02**)

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).

It's an error if:

- Y and Z are both omitted
The axis words are all optional except that at least one of Y and Z must be used.
- J and K are both omitted
J and K are optional except that at least one of the two must be used.



EXAMPLE

G17 G02 X1.0 Y1.6 I0.3 J0.4 Z0.9 is a center format command to mill an arc in incremental arc distance mode (**G91.1**) that makes 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's 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
```

10.2.5 Dwell (G04)

For a dwell, program: G04 P~

- P~ is the 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)

Troubleshooting

It's an error if:

- The P number is negative

10.2.6 Set Offsets (G10)

Use the controls on the Offsets tab to set offsets. You can program offsets with the G10 G-code command.

Read the following sections for reference:

Set Tool Table (G10 L1).....	185
Set Coordinate System (G10 L2).....	185
Set Tool Table (G10 L10).....	185
Set Tool Table (G10 L11).....	186
Set Coordinate System (G10 L20).....	186

Set Tool Table (G10 L1)

To define an entry in the tool table, program: G10 L1 P~ R~

- P~ is the tool number
- R~ is the 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.

Troubleshooting

It's 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

Set Coordinate System (G10 L2)

To define the origin of a work offset coordinate system,

program: G10 L2 P~ <axes R~>

- P~ is the number of coordinate system to use (G54 = 1, G59.3 = 9)
- R~ is the rotation about the rotation about the Z-axis

The G10 L2 P~ command doesn't change from the current coordinate system to the one specified by P. Use G54 through G59.3 to select a coordinate system.

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's currently active, the new coordinates take effect immediately. For example, if a G92 origin offset was in effect before G10 L2, it continues to be in effect after.

Optionally program R to indicate the rotation of the XY axis around the Z-axis. The direction of rotation is counterclockwise (viewed from the positive end of the Z-axis). When a rotation is in effect, jogging an axis only moves that axis in a positive or a negative direction — not along the rotated axis. To cancel a rotation for the active coordinate, program G10 L2 P0 R0.

Troubleshooting

It's an error if:

- The P number does not evaluate to an integer in the range 0-500
- An axis other than X or Z is programmed

Set Tool Table (G10 L10)

To change the tool table entry for tool P so that if the tool offset is reloaded with the machine in its current position and with the current G5x and G92 offsets active, program: G10 L10 P~ R~

10: PROGRAMMING

10.2 Programming G-Code

- $P\sim$ is the tool number
- $R\sim$ is the 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 (`G38`).

Troubleshooting

It's 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

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's set so that the current coordinates would become the given value if the new tool offset is reloaded and the machine is placed in the `G59.3` coordinate system without any `G92` offset active. This allows you to set the `G59.3` coordinate system according to a fixed point on the machine, and then use that fixture to measure tools without regard to other currently active offsets.

Program: `G10 L11 P~ X~ Y~ Z~ R~`

- $P\sim$ is the tool number
- $R\sim$ is the radius of tool

Troubleshooting

It's 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

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~`

- $P\sim$ is the number of coordinate system to use (`G54 = 1`, `G59.3 = 9`)
- $X\sim$ is the X-axis coordinate

- $Y\sim$ is the Y-axis coordinate
- $Z\sim$ is the Z-axis coordinate
- $A\sim$ is the A-axis coordinate

Troubleshooting

It's an error if:

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

10.2.7 Plane Selection (G17, G18, G19)

To select the XY-plane as active, program: `G17`

To select the XZ-plane as active, program: `G18`

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` through `G89`) is interpreted.

10.2.8 Length Units (G20 and G21)

To set length units to inches, program: `G20`

To set length units to millimeters, program: `G21`



Tip! Program either `G20` or `G21` near the beginning of a program, before any motion occurs. Avoid using either one anywhere else in the program. It's your responsibility to make sure that all numbers are appropriate for use with the current length units.

10.2.9 Return to Predefined Position (G28 and G28.1)

To make a rapid linear move from the current position to the absolute position of the values in parameters 5161-5166: `G28`

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



Note: Any axis not specified won't move.

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 machine coordinates in the native machine units of inches.

To store the current absolute position into parameters 5161-5163, program: `G28.1`

Troubleshooting

It's an error if:

- Cutter Compensation is turned on

10.2.10 Return to Predefined Position (G30 and 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 machine coordinates in the native machine units of inches.

To make a rapid traverse move from the current position to the absolute position of the values in parameters, program: `G30`

To make a rapid traverse move to the position specified by axes including any offsets, then make a rapid traverse move to the absolute position of the values in parameters 5181 and/or 5183, program: `G30 X~ Z~`



Note: Any axis not specified won't move.

To store the current absolute position into parameters 5181-5183, program: `G30.1`

Troubleshooting

It's an error if:

- Cutter Compensation is turned on

10.2.11 Automatically Measure Tool Lengths with an ETS (G37 and G37.1)

Use G37 and G37.1 with an Electronic Tool Setter (ETS) to enable automatic length measurement. For automated use, add a G37 command after an M6 tool change commands. If you're using the ETS with a mill, the input port varies depending on your machine:

- **M Series Mills** Plug the ETS into the Accessory Input 2 port. You can still use the Accessory Input 1 port for other probes and accessories.
- **Older PCNC Mills** Plug the ETS into the (single) accessory input port.

Move to G37 Position Over ETS (G37.1)

To move to the G37 position (over the ETS), program: `G37.1`

To set the G37 position:

1. Jog the machine over the center of the ETS.
2. From the **Probe** tab, on the **ETS Setup** tab, select **Set G37**

ETS Position.

The read-only DROs in the **ETS G37 Position Setup** group display the new position.

The G37 position is in G53 machine coordinate space. It defaults to (0, 0, 0), or the top left rear of machine travel (the same as the X-, Y-, and Z-axis reference position). G37.1 supports X and Y tool offsets. If there are X or Y tool offsets present in the tool table (manually applied through a G10 L1 command), they offset the spindle position. This enables G37 for tools mounted in an auxiliary spindle installed on the spindle column.



Note: If G10 L1 is used to change the X or Y offset of the currently loaded tool, you must then apply the new offsets with a G43 command.

G37.1 performs as follows:

1. A rapid upward move to the Z clear position (which is always G53 Z = 0.0).
2. A rapid move in X and Y to the X and Y ETS coordinates.
3. A rapid downward move in Z to the Z ETS coordinate.



Note: The Z word saves time by rapidly moving closer to the ETS before the slower probing begins. You must use caution if you set this lower than G53 Z = 0.0. If you don't, there's a risk that a long tool could collide with the ETS and damage it and the tool.

Move and Measure Tool Length (G37)

To move and measure the tool length, program: `G37 H~ P~`

- `H~` saves the measured tool length to the H tool table entry instead of the current tool number's entry. You could use this to track tool wear between the two tool table entries, for example.



Note: The newly measured tool length isn't applied, but it's stored in the tool table entry for tool number H.

10: PROGRAMMING

10.2 Programming G-Code

- **P~** is positive or negative tolerance. It measures the tool length, but, instead of storing it in the tool table, compares it to the length in the tool table. If the difference exceeds the P tolerance, the G-code program stops.

You could use this to detect broken or improperly inserted tools that are not fully seated in the spindle, for example.

G37 with no optional words moves to the G37 ETS position (through G37.1), probes the ETS, stores the new tool length in the tool table entry of the current tool, and applies the tool length offset.

G37 fails if the spindle nose hasn't been referenced to the ETS after a Z-axis reference. This sets a G53 coordinate at the ETS trigger point such that the measured tool length is the distance of the spindle nose to ETS reference. For more information, see the **ETS G37 Spindle Nose Reference** group on the

ETS Setup tab.

So that tool length measurements have consistent results, G37 uses the fine probe feed rate of 2.5 in./min for the final ETS touch. G37 uses the rough probe feed rate for the first ETS touch.

G37 performs as follows:

1. Issues a G37.1 move to the ETS location.
2. A downward rough probe feed rate move until the tool triggers the ETS.
3. An upward retract move of 0.100 in. to back off the triggered ETS.
4. A downward slow ETS probe feed rate move until the tool triggers the ETS.
5. An upward retract move of 0.100 in. to back off the triggered ETS.
6. An upward rapid move to the G37 ETS Z position.

10.2.12 Straight Probe (G38.x)

G38 . 2 probes toward the workpiece, stops on contact, and signals error if failure

G38 . 3 probes toward the workpiece and stops on contact

G38 . 4 probes away from the workpiece, stops on loss of contact, and signals error if failure

G38 . 5 probes away from the workpiece and stops on loss of contact

G38 . 6 moves away from the workpiece and ignores probe input

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

Conventionally, the probe is tool #99. The rotational axis words are allowed, but it's 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 tool in the spindle must be a probe.

In response to this command, the machine 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.

Troubleshooting

It's an error if:

- The current point is less than 0.01 in. (0.254 mm) 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
The linear axis words are optional, except that at least one of them must be used.
- Feed rate is zero
- The probe is already tripped

Use the Straight Probe Command

When you use 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, you may be able to find:

- Without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane
- If the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane
- 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

If the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, you can use

the straight probe command for things like 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
```

10.2.13 Cutter Compensation (G40, G41, G42)

To turn Cutter Compensation off, program: G40

It's okay to turn compensation off when it is already off.

It's 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

To program Cutter Compensation to the left of the programmed tool path (as viewed looking down on the machine), program: G41 D~

To program Cutter Compensation to the right of the programmed tool path (as viewed looking down on the machine), program: G42 D~

- D~ is the 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.

The lead in move must be at least as long as the tool radius.

The lead in move can be a rapid move.

It's an error if:

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

10.2.14 Dynamic Cutter Compensation (G41.1 and 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~

- D~ is the tip radius multiplied by two

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.

Troubleshooting

It's an error if:

- Cutter Compensation is commanded to turn on when it is already on

10.2.15 Apply Tool Length Offset (G43)

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

10: PROGRAMMING

10.2 Programming G-Code

- **H~** is the tool number associated with the length offset to be applied.



Note: Generally speaking, the value of the **H~** word should match the active tool number (**T~** word).

It's okay to program using the same offset already in use, or to program without a tool length offset (if none is currently being used).

Troubleshooting

It's an error if:

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

10.2.16 Engrave Sequential Serial Number (G47)

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

- **Z~** is the depth of cut of the engraving
- **R~** is the retract height between character segments in the numbers
- **X~** is, if present, the starting X position, or the left side of the serial number
If omitted, the current X position is assumed.
- **Y~** is, if present, the starting Y position, or the bottom side of the serial number
If omitted, the current Y position is assumed.
- **P~** is, if present, the X extent (width) in current units (inches or millimeters) of the engraved number
- **Q~** is, if present, the Y extent (height) in current units (inches or millimeters) of the engraved number
- **D~** is, if present, the requested number of decimals of the engraved number
If the requested D value exceeds 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.



EXAMPLE

A serial number of 10, where D = 4, engraves as 0010. A serial number of 9056, where D = 3, engraves as 9056.

Troubleshooting

It's 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)

10.2.17 Cancel Tool Length Compensation (G49)

To cancel tool length compensation, program: **G49**

10.2.18 Absolute Coordinates (G53)

For rapid linear motion to a point expressed in absolute coordinates, program: **G01 G53 X~ Y~ Z~** (or use with **G00** instead of **G01**)

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** isn't 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 machine won't go that fast). If **G00** is active, the speed of motion is the current traverse rate (or slower if the machine won't go that fast).

Troubleshooting

It's an error if:

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

10.2.19 Select Work Offset Coordinate System (G54 to G54.1 P500)

You can save up to 500 work offsets in PathPilot. The naming structure varies based on the offset number.

- To select work offset 1, program: **G54** or **G54.1 P1**
- To select work offset 2, program: **G55** or **G54.1 P2**
- To select work offset 3, program: **G56** or **G54.1 P3**

- To select work offset 4, program: `G57` or `G54.1 P4`
- To select work offset 5, program: `G58` or `G54.1 P5`
- To select work offset 6, program: `G59` or `G54.1 P6`
- To select work offset 7, program: `G59.1` or `G54.1 P7`
- To select work offset 8, program: `G59.2` or `G54.1 P8`
- To select work offset 9, program: `G59.3` or `G54.1 P9`
- To select a work offset beyond the standard 9 (listed above), program: `G54.1 P###`, where `P###` is a parameter indicating the index of the work offset you want to use (work offset 10 through work offset 500).



EXAMPLE

To select the 124th work offset, program `G54.1 P124`.

For information, see "About Work Offsets" (page 168).

Troubleshooting

It's 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 (i.e., System 1 X=5221, Z=5223; System 2 X=5141, Z=5143; up to System 9 X= 5381, Z = 5383).

10.2.20 Set Exact Path Control Mode (G61)

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

10.2.21 Set Blended Path Control Mode (G64)

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

- `P~` is, if present, 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~` is, if present, the maximum deviation from collinearity that will collapse a series of linear `G01` moves at the same feed rate into a single linear move.

It's okay to program for the mode that is already active.

10.2.22 Distance Mode (G90 and G91)

Interpretation of the operating system code can be in one of two distance modes: absolute or incremental.

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.

10.2.23 Arc Distance Mode (G90.1 and 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/G03` 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.

10.2.24 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3)



IMPORTANT! This is a legacy feature. Most modern programming methods don't use temporary work offsets.

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

- `X~` is the X-axis coordinate
- `Y~` is the Y-axis coordinate
- `Z~` is the Z-axis coordinate
- `A~` is the A-axis coordinate

`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` through `G59.3`).

10: PROGRAMMING

10.2 Programming G-Code

EXAMPLE

If the current controlled point is at $X = 4$, and there is currently no G92 offset active, and 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's an error if:

- All axis words are omitted

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

To reset axis offsets to zero and sets parameters 5211 - 5219 to zero, program: G92 . 1

To reset axis offsets to zero, program: G92 . 2

To set the axis offset to the values saved in parameters 5211 to 5219, program: G92 . 3

10.2.25 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 **[1/F number]** minutes.

EXAMPLE

If the F number is 2.0, the move should be completed in half a minute.

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.

Troubleshooting

It's 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

10.2.26 Spindle Control Mode (G96 and G97)

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

- D~ is the maximum spindle RPM.
This word is optional.
- S~ is the surface speed.



Note: If G20 is the active mode, the value is interpreted as feet per minute. If G21 is the 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).

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

To set RPM mode, program: G97

Troubleshooting

It's an error if:

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

10: PROGRAMMING

10.3 Programming Canned Cycles

10.3 PROGRAMMING CANNED CYCLES

Read the following sections for reference:

10.3.1 Canned Cycles Reference	194
10.3.2 High Speed Peck Drill (G73)	195
10.3.3 Cancel Canned Cycles (G80)	195
10.3.4 Drilling Cycle (G81)	195
10.3.5 Simple Drilling Cycle (G82)	197
10.3.6 Peck Drilling Cycle (G83)	197
10.3.7 Boring Cycle (G85)	197
10.3.8 Boring Cycle (G86)	197
10.3.9 Boring Cycle (G88)	197
10.3.10 Boring Cycle (G89)	197

10.3.1 Canned Cycles Reference

Supported Canned Cycles

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 we use assume the XY-plane has been selected. The behavior is always analogous if the YZ- or XZ-plane is selected.

- $X\sim$ is the X-axis coordinate
- $Y\sim$ is the Y-axis coordinate
- $Z\sim$ is the Z-axis coordinate
- $A\sim$ is the A-axis coordinate
- $R\sim$ is the 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)

- $L\sim$ is the 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's 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 — it keeps its value on subsequent blocks if they're 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 — 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.

Troubleshooting

It's 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's 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's 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's 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.

10.3.2 High Speed Peck Drill (G73)

The G73 cycle is intended for deep drilling with chip breaking. The retracts in this cycle break the chip but do not totally retract the drill from the hole. It's suitable for tools with long flutes which clear the broken chips from the hole.

Program: G73 X~ Z~ R~ L~ Q~

- Q~ is the delta increment along the Z-axis

The G73 cycle is as follows:

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

Troubleshooting

It's an error if:

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

10.3.3 Cancel Canned Cycles (G80)

To cancel all canned cycles, program: G80

It's okay 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."**

10.3.4 Drilling Cycle (G81)

The G81 cycle is intended for drilling.

Program: G81 X~ Y~ Z~ A~ R~ L~

The G81 Cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Retract the Z-axis at traverse rate to clear Z.

Examples

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

10: PROGRAMMING

10.3 Programming Canned Cycles

EXAMPLE

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

```
G90 G81 G98 X4 Y5 Z1.5 R2.8
```

This means that it's in absolute distance mode (G90), old Z retract mode (G98) and the G81 drilling cycle is 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:

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

The current position is (1, 2, 3), the XY-plane has been selected, the following line of code is interpreted: **G91**

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

This means that it's in incremental distance mode (G91), old Z retract mode, and the G81 drilling cycle is 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:

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):

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):

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
```

10.3.5 Simple Drilling Cycle (G82)

The G82 cycle is intended for drilling.

Program: G82 X~ Y~ Z~ A~ R~ L~ P~

The G82 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Dwell for the P number of seconds.
- × **Step 4:** Dwell for the P number of seconds.

10.3.6 Peck Drilling Cycle (G83)

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum).

Program: G83 X~ Y~ Z~ A~ R~ L~ Q~

- Q~ is a delta increment along the Z-axis

The G83 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
- × **Step 3:** Rapid back out to the clear Z.
- × **Step 4:** Repeat Steps 1 through 3 until the Z position is reached at Step 1.
- × **Step 5:** Rapid back down to the current hole bottom, backed off a bit.
- × **Step 6:** Retract the Z-axis at traverse rate to clear Z.

Troubleshooting

It's an error if:

- The Q number is negative or zero

10.3.7 Boring Cycle (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~

The G85 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Retract the Z-axis at the current feed rate to clear Z.

10.3.8 Boring Cycle (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~

The G86 cycle is as follows:

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

Troubleshooting

It's an error if:

- The spindle is not turning before this cycle is executed

10.3.9 Boring Cycle (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~

The G88 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Dwell for the P number of seconds.
- × **Step 4:** Stop the spindle turning.
- × **Step 5:** Stop the program so the operator can retract the spindle manually.
- × **Step 6:** Restart the spindle in the direction it was going.

10.3.10 Boring Cycle (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~

10: PROGRAMMING

10.3 Programming Canned Cycles

The G89 cycle is as follows:

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

10.4 PROGRAMMING M-CODE

Read the following sections for reference:

10.4.1 Supported M-Codes Reference	199
10.4.2 Program Stop and Program End (M00, M01, M02, and M30)	199
10.4.3 Spindle Control (M03 and M05)	200
10.4.4 Tool Change (M06)	200
10.4.5 Coolant Control (M07, M08, and M09)	201
10.4.6 Override Control (M48 and M49)	201
10.4.7 Feed Override Control (M50)	201
10.4.8 Spindle Speed Override Control (M51)	201
10.4.9 Set Current Tool Number (M61)	201
10.4.10 Set Output State (M64 and M65)	201
10.4.11 Wait on Input (M66)	201

10.4.1 Supported M-Codes Reference

M-Code	Description
<u>M00</u>	Program stop
<u>M01</u>	Optional program stop
<u>M02</u>	Program end
<u>M03</u>	Rotate spindle clockwise
<u>M05</u>	Stop spindle rotation
<u>M07</u> , <u>M08</u>	Coolant on
<u>M09</u>	All coolant off
<u>M30</u>	Program end and rewind
<u>M48</u>	Enable speed and feed override
<u>M49</u>	Disable speed and feed override
<u>M64</u>	Activate output relays
<u>M65</u>	Deactivate output relays
<u>M66</u>	Wait on an input
	 Note: M64 through M66 is only useful with a <u>USB M-Code I/O Interface Kit (PN 32616)</u>.
<u>M98</u>	Call subroutine

M-Code	Description
<u>M99</u>	Return from subroutine/repeat
<u>M301</u> , <u>M302</u> , <u>M303</u>	USB camera control

10.4.2 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's okay 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, selecting Cycle Start 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:

- 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. Selecting Cycle Start starts the program back at the beginning of the file.

Display Information and Capture Images During an M00 or M01 Break

Display Information with Images

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

10: PROGRAMMING

10.4 Programming M-Code

image in the Tool Path display when the program reaches the M00 or M01 break.

To display an image during an M00 or M01 break:

1. Move an image file with a .jpg or .png extension to the PathPilot controller in one of the following locations:
 - In the same folder as the G-code program
 - In an images folder within the G-code program's folder
 - In an images folder within the home directory
2. Program an M00 or M01 break, and, using parentheses, type the full file name of the image (including its extension).



EXAMPLE

M01 (Op1_Setup.jpg) displays the image file on the Tool Path display.

3. The image file displays on the Tool Path display.

Display Information with Text

To display a message on the Tool Path display:

1. Program an M00 or M01 break, and, using parentheses, type a message that you'd like to display on the screen.



EXAMPLE

M01 (Check coolant nozzles are pointed correctly) displays *Check coolant nozzles are pointed correctly* across the bottom of the Tool Path display.

2. The message displays on the Tool Path display.

Capture Images with a USB Camera

In addition to displaying information like pictures or messages during an M01 break, you can also use a USB camera (if installed) to take a picture.

To use M01 to take pictures:

1. Add M01 (op1_setup.jpg) into your G-code program.
2. Run the G-code program.
3. When PathPilot executes the M01 it looks to see if the comment contains a file name.
 - If there isn't a file name: The comment is shown as instructional text across the tool path.

- If there is a file name, but the file doesn't exist yet and the extension is .jpg, .png, or .jpeg: The USB cameras are initialized and shown in the tool path display.

4. Select the **Shutter** button to take the picture and create the op1_setup.jpg file.

In future runs of the G-code program, **op1_setup.jpg** will display to the operator for instructional purposes on the workpiece.

For more information, see "Use a USB Camera" (page 119).

10.4.3 Spindle Control (M03 and M05)

To start the spindle turning clockwise at the currently programmed speed, program: M03

The speed is programmed by the S word.

To stop the spindle from turning, program: M05

It's okay to use M03 if the spindle speed is set to 0; if this is done, the spindle won't start turning. If later the spindle speed is set above 0, 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.

10.4.4 Tool Change (M06)

To execute a tool change sequence, program: M06

M06 behaves differently depending on whether or not the machine is equipped with an Automatic Tool Changer (ATC):

- If you have 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, you're prompted to manually change the tool and select Cycle Start to confirm the tool change. This resumes the program.
- If you don't have an ATC:
 - M06 commands the machine, stops the spindle, pauses program execution, and prompts operator to change tools by flashing Tool Change LED.
 - The program resumes after you select Cycle Start to confirm that the tool has been changed.

We recommend putting the T~, the M06, and the G43 H~ on one line (block) of code.



EXAMPLE

N191 M06 T3 G43 H3

10.4.5 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

It's always okay to use any of these commands, regardless of what coolant is on or off.

10.4.6 Override Control (M48 and M49)

To enable the speed and feed override, program: M48

To disable both overrides, program: M49

It's okay to enable or disable the switches when they are already enabled or disabled.

10.4.7 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).

10.4.8 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.

10.4.9 Set Current Tool Number (M61)

To change the current tool number while in MDI or manual mode, program: M61 Q~

- Q~ is the tool number

Troubleshooting

It's an error if:

- Q~ is not 0 or greater

10.4.10 Set Output State (M64 and M65)



Note: These commands are only useful when the machine is equipped with the USB M-Code I/O Interface Kit.

There are four output relays available on the USB I/O module.

To activate output relays (contact close), program: M64

To deactivate output relays (contact open), program: M65

There are four contacts, numbered from 0 to 3. The contact is specified by the P word.



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.



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
```

10.4.11 Wait on Input (M66)



Note: This command is only useful when the machine is equipped with the USB M-Code I/O Interface Kit.

There are four digital inputs available on the USB I/O module.

M66 P- | E- <L->

10: PROGRAMMING

10.4 Programming M-Code

- **P**– is the digital input number from 0 to 3.
- **L**– is the wait mode type:
 - 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**– is 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.

10.5 PROGRAMMING INPUT CODES

Read the following sections for reference:

10.5.1 Feed Rate (F)	203
10.5.2 Spindle Speed (S)	203
10.5.3 Change Tool Number (T)	203

10.5.1 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.

10.5.2 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's okay 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's okay to program S0, but the spindle does turn if that is done.

Troubleshooting

It's an error if:

- The S number is negative

10.5.3 Change Tool Number (T)

It's your responsibility to make sure that the machine is in a safe place for changing tools (for example, by using G30). This allows optimization of motion which can save time. You can provide a pause for manual intervention with `M00` or `M01` before the tool change.

Troubleshooting

It's an error if:

- A negative T number is used
- A T number larger than 1000 is used

10: PROGRAMMING

10.6 Advanced Programming

10.6 ADVANCED PROGRAMMING

Parameter and expression programming language features are not used in common G-code application (hand coding), G-code created by PathPilot conversational programming, or the majority of third-party CAM-programming systems.

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. We don't recommend writing parametric G-code — it's difficult to debug, and difficult for another operator to understand.

Modern CAM virtually eliminates the need for it.

Read the following sections for reference:

10.6.1 Parameters	204
10.6.2 Expressions	206
10.6.3 Subroutines	207

10.6.1 Parameters

Read the following sections for reference:

Parameters Reference	204
Numbered Parameters Reference	205
Subroutine Parameters Reference	205
Named Parameters Reference	205

Parameters Reference

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 Use

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.

Numbered Parameters Reference

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 lathe. The remaining parameters for unused axes are undefined.

Read-Only Parameters

- 1-30: Subroutine local parameters of call arguments. These parameters are local to the subroutine. For further information, see Programming with Subroutines 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
- 5420-5428: Current position including offsets in current program units (X Y Z A B C U V W)

Subroutine Parameters Reference

Subroutine parameters are specifically reserved for call arguments. By definition, these are parameters #1-#30 and are local to the subroutine.

Named Parameters Reference

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

10: PROGRAMMING
10.6 Advanced Programming

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.
The global parameters _a, _b, _c, ... _z are reserved for special use. Do not use these parameters.

EXAMPLES

- #<_endmill_dia> = 0.049 is a declaration of named global variable.
- #<_endmill_rad> = [#<_endmill_dia>/2.0] is a reference to previously declared global variable.
- o100 call [0.0] [0.0] [#<_inside_cutout>-#<_endmill_dia>] [#<_Zcut>] [#<_feedrate>] is mixed literal and named parameters.

10.6.2 Expressions

An expression is a set of characters starting with a left bracket ([) and ending with a 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.
Read the following sections for reference:

Binary Operators Reference206
Functions Reference. 206

Binary Operators Reference

Binary operators only appear inside expressions. There are three types of binary operators: mathematical, logical, and relational.

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 as follows, from highest to lowest:

- 1. **
- 2. * / MOD
- 3. + -
- 4. EQ NE GT GE LT LE
- 5. AND OR XOR

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.

Functions Reference

- The available functions are:
- 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 (ex)

- `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

10.6.3 Subroutines

Subroutines are subprograms that are called from inside another program.

Read the following sections for reference:

Subroutines Reference	207
Conditional Subroutines Reference	208
Repeating Subroutines Reference	208
Looping Subroutines Reference	208

Subroutines Reference

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:

- `O123`
- `O99999`
- `O<my test code>`

Subroutine labels may be used in any order, but they 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:

- `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

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.

Example of `sub`, `endsub`, `call`:

```
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 example, 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 as long as:

- The file is named the same as your call.
- The file includes a `sub` and `endsub` in the file.
- The file is in the directory `/subroutines`.
- The file name only includes lowercase letters, numbers, dashes, and underscores.
- The file only contains a single subroutine definition.
- The file ends with the extension `.nc`.

10: PROGRAMMING

10.6 Advanced Programming

Note: File names are lowercase letters only.

 `o<MyFile>` is converted to `o<myfile>` by the interpreter.

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`

`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: `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.

Conditional Subroutines Reference

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, then 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
```

Repeating Subroutines Reference

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)
```

Looping Subroutines Reference

Subroutines can be looped using the do/while or while/endwhile keyword constructs.

do/while

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
```

- 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

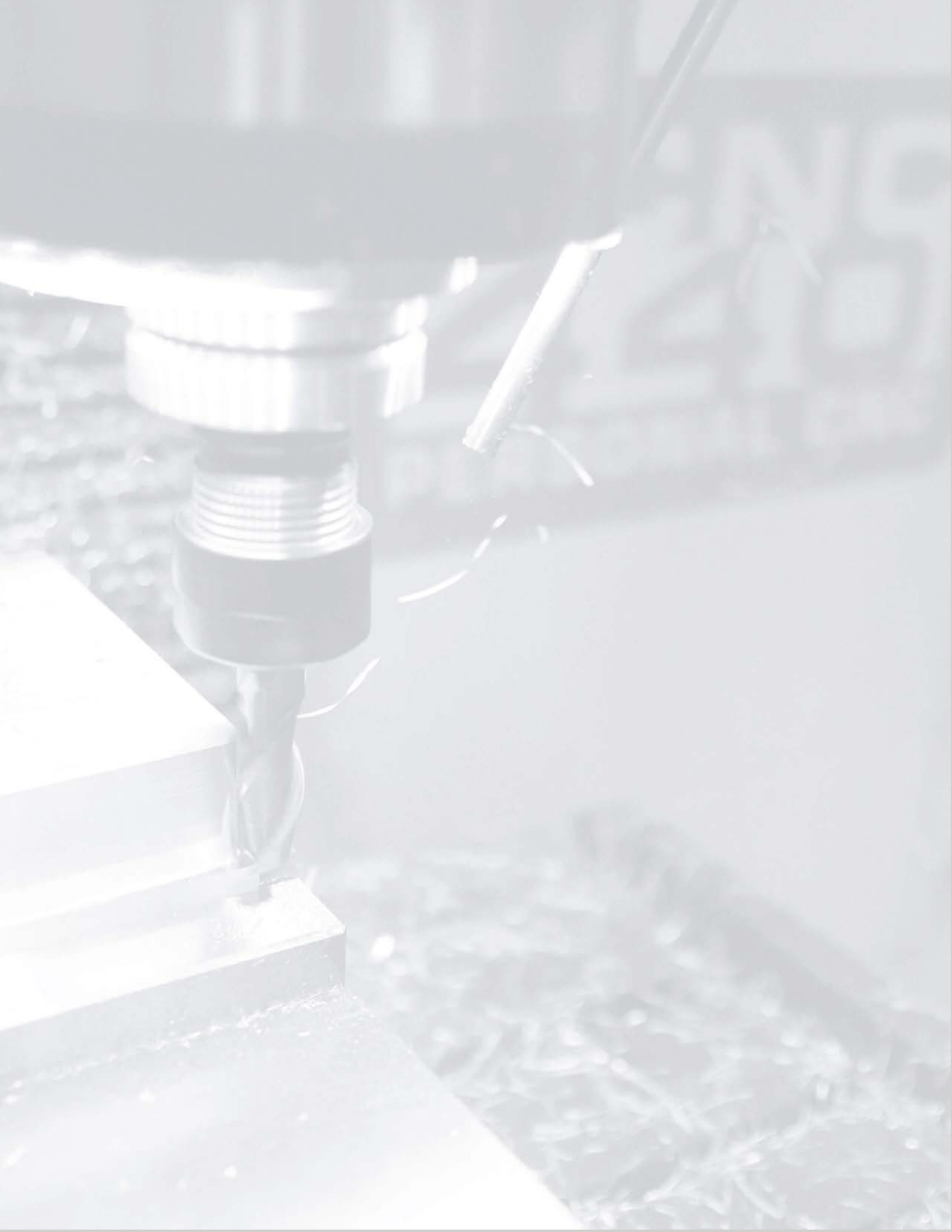
while/endwhile

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



MACHINE MAINTENANCE

IN THIS SECTION, YOU'LL LEARN:

- About the required maintenance procedures that you must do so that this machine operates as designed.



Before operating the machine in any way, you must read and understand this section.

CONTENTS

11.1 Maintenance Safety	212
11.2 Maintenance Schedules	213
11.3 Regular Maintenance	214
11.4 Advanced Maintenance Adjustments	217
11.5 Transporting the Machine	222

11: MACHINE MAINTENANCE

11.1 Maintenance Safety

11.1 MAINTENANCE SAFETY

Read and understand the following safety messages before beginning any maintenance procedures.

11.1.1 All Maintenance Procedures

- ✓ Understand that the machine is automatically controlled and can start at any time.
- ✓ Power off the machine and disconnect the pneumatic supply before doing any maintenance procedures.
- ✓ When appropriate, lockout/tagout the Main Disconnect switch and the pneumatic supply line before doing any maintenance procedures.
- ✓ Wear safety eye protection rated for ANSI Z87+.

11.1.2 Swarf Maintenance Procedures

- ✓ Wear work gloves.

11.2 MAINTENANCE SCHEDULES

To keep your machine running as smoothly as possible, you must regularly do the following maintenance procedures.



Note: Before you begin any maintenance procedures, read and understand "Maintenance Safety" (on the previous page).

If you disassemble any components, refer to the machine's reference drawings when you've completed the maintenance procedure. For information, see "Diagrams and Parts Lists" (page 223). For any additional support, we can help. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.

11.2.1 Daily

- ☐ Clean the way covers of chips.
- ☐ Examine the way oil level and refill as necessary.
- ☐ Retract the plunger on the manual oiler, hold it in the retracted position for two seconds, and then release it.



Note: The plunger should move slowly from the retracted position. If it doesn't, it could indicate a malfunctioning oiler.

- ☐ If you have pneumatic accessories installed (like a Power Drawbar), verify that the regulator has at least 90 psi compressed air.
- ☐ Verify that the door switches are properly functioning.
- ☐ Spray exposed, non-painted metal surfaces with WD-40® (or similar water displacement oil) to prevent rust.

11.2.2 Weekly

- ☐ Clean all exterior surfaces with a clean rag.
- ☐ Verify that the machine's lubrication points (sliding surfaces and ball screw nuts) are receiving way oil.
- ☐ Examine the enclosure and other guards for damage.
- ☐ Examine the drawbar for wear.
- ☐ Examine the Tormach Tooling System (TTS) collet for wear.
- ☐ Remove vises, fixture plates, rotary tables, and other

accessories, and then examine them for rust. If needed, reapply rust inhibitor before reinstalling workholding and accessories.

- ☐ Check coolant concentration.

11.2.3 Monthly

- ☐ Clean the electrical cabinet vents of dust with a clean cloth or compressed air.
- ☐ Clean fine swarf from the coolant tank.
- ☐ Examine all axes' slideway surfaces and ball screws to determine if an oil film is present.
- ☐ Examine the air tool oil level in the FRL Filter-Regulator-Lubricator and refill as necessary.

11.2.4 Semi-Annually

- ☐ Examine the spindle belt for wear.
- ☐ Examine the way covers for wear.
- ☐ Examine the X-axis flex conduit for wear.
- ☐ Inspect the oil system for blockages. Clean or replace any components as needed.
- ☐ Check the lubrication hoses for signs of wear or cracking.
- ☐ Verify that the limit switches are properly functioning.
- ☐ Remove the way covers and examine them for wear. Clean any chips or oil underneath the way covers.
- ☐ Examine the windows and replace them if necessary.

11: MACHINE MAINTENANCE

11.3 Regular Maintenance

11.3 REGULAR MAINTENANCE

- 11.3.1 Clean the Way Covers..... 214
- 11.3.2 Clean the Coolant System..... 214
- 11.3.3 Examine the Drawbar..... 215
- 11.3.4 Examine the Enclosure Windows..... 215
- 11.3.5 Lubricate the Machine..... 215
- 11.3.6 Examine the Spindle Belt 216
- 11.3.7 Examine the Tormach Tooling System (TTS) Collet..... 216
- 11.3.8 Prevent Rust 216

11.3.1 Clean the Way Covers

To extend the service life of the machine, you must make sure that the way covers are protecting the machine's slideways. Regularly do the following:

- Examine the way covers for signs of wear or damage. If the way covers are damaged, you must replace them.
- Use a chip brush to clean the way covers of chips and other swarf.

11.3.2 Clean the Coolant System

To extend the service life of the coolant and the coolant pump, you must make sure that both are free of contaminants. Regularly do the following:

- Use a chip brush to clean the machine, chip trays, and enclosure of chips and other swarf.
- Empty the chip strainer on the coolant tank.
- Measure the coolant concentration, pH, and volume, and then compare your findings to the material's safety data sheet (SDS).
 - **Coolant Level Too Low** Fill the coolant tank with fresh coolant, making sure that it is the appropriate concentration.
 - **Coolant pH Outside of Manufacturer's Recommended Coolant Limits** Adjust the pH according to the manufacturer's recommendation.



Note: You must only use coolant that is recommended for machining. See "Cutting Fluid Reference" (below).

- Examine the coolant pump's impeller for blockages.
- To remove tramp way oil from the coolant tank, use one of the following:

- Oil Skimmer Kit for 1100M/1100MX, 770M/770MX, and PCNC 440 (PN 39298)
- Floating Tramp Oil Collection Pillow (PN 31925)

Periodically do the following:

- Remove the coolant tank covers and the chip strainer, and then clean the fine swarf from the coolant tank.
- If you're using a filter to prevent small particles from entering the coolant tank, replace it when it's saturated with chips, swarf, or tramp oil.
- Examine the coolant for contamination, and determine if the coolant's oil-water emulsion has broken.
- Drain and thoroughly clean the coolant tank, and then replace with fresh coolant.

NOTICE! Hazardous Waste: Only dispose of coolant as advised by local environmental protection authorities. You must not dispose of coolant in a septic system or a sewer.

About Cutting Fluid

Clean and well maintained coolant has increased performance, prevents rust, and smells better than contaminated coolant. Coolant becomes contaminated when:

- Swarf accumulates in the chip pans, the chip strainer, or the coolant tank.
- Tramp oil isn't regularly removed from the coolant tank.
- The concentration or pH aren't maintained to the manufacture's specifications.

Cutting Fluid Reference

Using flood coolant provides the following benefits:

- Cools the cutting zone
- Flushes away swarf
- Lubricates the cutting tool
- Provides some degree of rust inhibition

You must only use flood coolant that is recommended for CNC machining applications, like:

- Soluble oils
- Semi-synthetic coolants
- Synthetic coolants

You must never use:

- Coolant without rust inhibitors
- Flammable coolants such as alcohols, diesel fuel, or kerosene



WARNING! Fire Hazard: The machine and its enclosure are not designed to contain fire or explosions. Only use materials and coolants that are intended for the specific machining operation. Never use flammable or explosive items.

- Straight cutting oil
- Water

If you use a coolant concentrate, you must mix it with water to the dilution ratio recommended by the coolant manufacturer. Use an optical refractometer to measure coolant concentration.

11.3.3 Examine the Drawbar

To extend the service life of the drawbar, you must make sure that the drawbar isn't damaged, and that it's properly lubricated.

Regularly do the following:

- Examine the drawbar for signs of wear or damage.
- Use Anti-Seize (PN 31273) to lubricate the following items:
 - Drawbar threads
 - Thrust shoulder of the drawbar
 - Drawbar's thrust washer

If you have an (optional) Power Drawbar, regularly do the following:

- Use Anti-Seize to lubricate the Power Drawbar disc springs (the Belleville washers) and clamping mechanism.
- Tighten the drawbar until it is as tight as possible while still releasing a Tormach Tooling System (TTS) tool holder using 90 psi (620 kPa) compressed air.

11.3.4 Examine the Enclosure Windows

Regularly examine the enclosure windows, and all other guarding, for signs of wear or damage.

When required, replace the windows with the following parts:

- PCNC 440 Enclosure, Side Window (PN 51460)
- PCNC 440 Enclosure, Left Door Window (PN 51453)

- PCNC 440 Enclosure, Right Door Window (PN 51454)

11.3.5 Lubricate the Machine

To keep the machine operating properly, and to extend the service life of the machine, you must verify that the slideway surfaces and ball screws are receiving lubrication.

Regularly do the following:

- Clean the oil reservoir's cover and all exposed surfaces surrounding the oil reservoir. Swarf or debris that enters the oil reservoir can plug the small, internal passages of the lubrication system.



Note: The strainer at the top of the oil reservoir isn't a filter, and can't remove all types of debris that may cause blockages.

- Examine all axes' slideway surfaces and ball screws to determine if an oil film is present. If there's not an oil film, examine the lubrication system for blockages, broken fittings, or pinched oil lines.
- Examine the amount of available oil and, if necessary, refill with new, high-quality ISO VG 68 grade Machine Oil (PN 31386).

About the Manual Oiler

The manual oiler operates with a plunger:

- **On the Pull Stroke** The oiler pulls machine oil from the oil reservoir.
- **On the Retract Stroke** The oiler pushes machine oil through the lubrication system.

The plunger is spring-loaded, and creates a light hydraulic pressure sufficient to distribute the oil through the machine. Do not push the plunger after you retract it. If at some point the manual oiler seems much easier on the push stroke, inspect the oil lines for damage.

When the machine is first powered on (and then after every four hours of operation), retract the oiler's plunger and hold it in the retracted position for two seconds. Then, release the oiler's plunger.



Note: The plunger should move slowly from the retracted position. If it doesn't, it could indicate a malfunctioning oiler.

11: MACHINE MAINTENANCE

11.3 Regular Maintenance

After a long period of inactivity, the lubrication system may need several pumps, as oil may have drained from the Z-axis lines.

You will have a more uniform distribution of oil if the machine is moving when the hydraulic pressure is applied. 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 machine as intended. Instead, the oil will squirt out at the point where the oil channel is exposed.

Lubrication System Reference

The machine's lubrication system distributes oil to the following 15 points:

- 12 slideways bearing surfaces
 - Four on the X-axis
 - Four on the Y-axis
 - Four on the Z-axis
- Three ball screw nuts

11.3.6 Examine the Spindle Belt

To extend the service life of the spindle belt, you must regularly examine it for signs of damage or wear.

If the spindle belt is damaged, you must replace it with Belt: PCNC 440 Spindle (PN 35799).

11.3.7 Examine the Tormach Tooling System (TTS) Collet

To extend the service life of the Tormach Tooling System (TTS) collet, you must regularly do the following:

- Examine the Tormach Tooling System (TTS) collet for signs of wear or damage.
- Make sure the inside of the Tormach Tooling System (TTS) collet is clean and dry.
- Use Anti-Seize (PN 31273) to lubricate the outside of the Tormach Tooling System (TTS) collet and the inside of the spindle taper.
- Use a mild degreaser to clean the shank of all tool holders.

11.3.8 Prevent Rust

Take proper care to protect all exposed iron and steel surfaces on your machine. To reduce the possibility of rust, you must regularly do the following:

- Clean the machine, chip pans, and enclosure of chips and other swarf with a chip brush.
- Clean all exterior surfaces with a mild cleaner.
- Only use flood coolant that is recommended for machining (see "Cutting Fluid Reference" (page 214)).
- Periodically test the coolant concentration, and adjust (if necessary) to meet coolant manufacturer's recommended concentration.
- Put LPS 3® (or similar rust inhibitor) on the machine before installing any workholding or accessories.
- Only operate the machine in a temperature- and humidity-controlled environment. Extreme changes in temperature or humidity can create condensation on the machine.
- Put LPS 3® (or similar rust inhibitor) on all exposed, non-painted metal surfaces before leaving the machine unused.
- Remove all workholding and accessories, and then check for rust.
- If needed, reapply rust inhibitor before reinstalling workholding or accessories.

If you find rust on the machine table, go to "Remove Rust" (below).

Remove Rust

Use a fine machinist's stone to remove light surface rust on the machine table: use light, even pressure, and move the stone in a random motion to avoid changing the flatness of the machine table.

If you find a lot of rust, or if there is rust on the machine's slideways or ball screws, take photos of all rust-covered areas. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.

11.4 ADVANCED MAINTENANCE ADJUSTMENTS



Note: Before you begin any maintenance procedures, read and understand "Maintenance Safety" (page 212).

- "About Advanced Maintenance" (below)
- "Definitions" (below)
- "How to Measure Lost Motion" (below)
- "Adjust the Gibs" (on the next page)
- "Adjust the Angular Contact Bearing Pre-Load" (page 220)
- "Adjust the Geometry of Precision Mating Surfaces" (page 220)
- "Adjust the Spindle Bearings" (page 221)

11.4.1 About Advanced Maintenance

Your machine is tuned at the factory to meet or exceed certain precision metrics, all of which are indicated on the included Certificate of Inspection.

Over the ownership lifetime of your machine, you may need to perform advanced maintenance adjustments to maintain the original factory precision. Before you do, read and understand the following sections:

- Gib adjustment
- Angular contact bearing preload adjustment
- Geometry adjustment (tram)

While the adjustments are generally used to address component wear-in over time, you can also use them to correct misalignment from misuse, a hard crash of the system, or component replacement. The frequency of these procedures depends on both how the machine is used and how often. Making the wrong adjustment could adversely affect mill precision. Before you making any advanced maintenance adjustment, make sure that you 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 decide to make an adjustment, be careful to avoid assuming the error. It could be attributed to a specific problem, or from the combined effect of several problems.



EXAMPLE

The Z-axis gib is too loose, which causes the spindle head to tilt slightly downward toward the column. It's easy to 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, though, this adjustment causes the column to slant back to correct for the head leaning down. Now, the machine is running in a slight parallelogram — and it still has a loose spindle head.

11.4.2 Definitions

The following definitions are important to the advanced maintenance discussion.

Lost Motion

The difference between commanded motion and observed motion, sometimes referred to as apparent backlash. There are a many components of lost motion, like:

- Bearing compressibility
- Conventional backlash
- Sliding friction
- Thermal expansion

Backlash

The major component of lost motion in a machine tool axis, which results from the clearance between moving mechanisms, sometimes referred to as play. There are two sources of conventional backlash that can be adjusted on the PCNC:

- The space between the gib and the way needed to support an oil film, which is tuned by tightening the gib.
- The space between the ball bearings and the races in the angular contact bearing pair that supports the ball screw, which is tuned by increasing the angular contact bearing pair preload.

11.4.3 How to Measure Lost Motion

Correctly measuring lost motion is critical to successfully making any advanced maintenance adjustments. Machine setup and tuning is done under no-load conditions; the precision measurements recorded in the Certificate of Inspection are taken under no-load conditions.

11: MACHINE MAINTENANCE

11.4 Advanced Maintenance Adjustments



Note: The accuracy of a machined feature isn't an indicator of machine precision. Many factors contribute to the overall machined-part accuracy, such as tool flex, workpiece flex, fixture flex, thermal expansion.

Required Tools

- Dial indicator (with increments less than 0.001 inches)
- Dial test indicator
- Magnetic dial stand

To Measure X-Axis Backlash

An analogous procedure is used to measure Y- and Z-axis backlash.

1. Mount a dial indicator on the machine table along the X-axis to the left of the spindle. Make sure that the tip points at the spindle.
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 the following image. Make sure that you're using the flat tip on the indicator, because it's contacting a curved surface.



Figure 11-1: Dial indicator positioning.

3. Slowly jog the X-axis in the positive direction until the indicator contacts the spindle.
4. Continue jogging the X-axis in the positive direction so that the dial makes at least one complete revolution. Stop jogging when the dial reads 0.
5. From the PathPilot interface, select **Zero X**.
6. In the MDI Line DRO field, type `G01 X.01 F5` to program a positive X move of 0.01 inches at a feed rate of 5 IPM.

The spindle head moves slightly in the +X direction. When finished, the indicator should read 0.010 inches.

7. In the MDI Line DRO field, type `G01 X0 F5` to program an X move back to 0. The spindle head moves slightly in the -X direction. When finished, the X DRO should display 0, and the dial indicator should read a number very close to 0. This value is the measured lost motion.

11.4.4 Adjust the Gibs

We recommend making a gib adjustment by measuring the axis lost motion while incrementally tightening the gibs to arrive at the correct setting. For information, see "About Gibs" (on the next page).



IMPORTANT! It's difficult to assess the correct clearance for the gib. A very small adjustment can create a dramatic change in sliding friction.

The gib tightening adjustment procedure for each linear axis is detailed in the following table.

NOTICE! Before making any adjustments to the Z-axis, you must make sure that you support the spindle head with a wooden block on the machine table.

Gib Plate	Tighten	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

Gib Plate	Tighten	Notes
Z-Axis	Upper screw clockwise/lower screw counterclockwise	
To loosen, reverse the rotation direction indicated in this table.		

To adjust the Z-axis gib:

Note: You can use a similar procedure to adjust the X-axis and Y-axis gibs. You must not make adjustments to the X- and Y-axis gib adjustments isolation — tightening or loosening a gib on either axis also has an effect on the opposing axis.

1. Make sure that the gib clearance is loose: loosen the upper gib screw eight rotations and tighten the lower gib screw eight rotations.
2. If you haven't yet done so, measure lost motion in the Z-axis (For information, see, go to "How to Measure Lost Motion" (page 217)).
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 machine, this value should be less than 0.002 inches on the Z-axis, and less than .0018 inches 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.
4. Re-measure the backlash.
5. Repeat Steps 1 through 4 until the measured backlash begins to increase. When finished, the gib setting should be slightly too tight.
6. Back the adjustment off to the point just before you saw the increased backlash.
This is the ideal setting for the axis.

Note: After any gib adjustment, make sure that both adjustment screws are tight to avoid moving the gib out of adjustment.

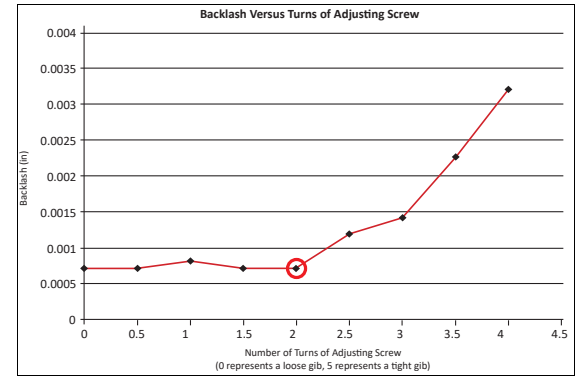


Figure 11-2: Backlash versus turns of adjusting screw graph.

About Gibs

The machine uses dovetail-gibbed ways to guide the X-, Y-, and Z-axis motion. Over time, the dovetail ways and gibs wear from sliding friction, and it may become necessary to tighten the gib to reduce axis backlash. To compensate for wear, the design of a dovetail-gibbed way allows for adjustment to the position of the gib in order to maintain an appropriate sliding clearance.

A properly adjusted gib minimizes lost motion by balancing conventional backlash and sliding friction:

- **If a Gib is Too Loose** It can result in excessive conventional backlash.
- **If a Gib is Too Tight** It can't adequately support an oil film, which can result 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. You can adjust these screws as a pair to tune the tightness and sliding friction of the dovetail way for each axis. For reference, see the Y-axis gib mechanical detail in the following image.

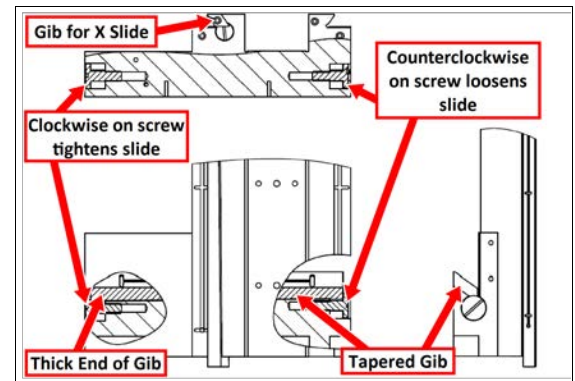


Figure 11-3: Y-axis gib mechanical detail.

11: MACHINE MAINTENANCE

11.4 Advanced Maintenance Adjustments



Note: The X- and Z-axis have similar detail.

11.4.5 Adjust the Angular Contact Bearing Pre-Load

For information, see "About Angular Contact Bearing Pre-Load" (below).

Required Tools

- Pin Spanner Wrench (PN 32655)

To Adjust the Angular Contact Bearing Pre-Load

1. Locate the adjustment nut on the back of each stepper motor.
2. Mark the current position of the adjustment nut on the back of the stepper motor.
3. Measure the lost motion in the axis. For information, see "How to Measure Lost Motion" (page 217).
4. Use a Pin Spanner Wrench to turn the adjustment nut clockwise 10 degrees.
5. Re-measure the lost motion in the axis.
6. Repeat Steps 4 to 5 until the lost motion value does not decrease from turning the adjustment nut.

NOTICE! Don't over-tighten the adjustment nut, which could cause the stepper motors to stall, or cause damage to the bearings and the motor.

About Angular Contact Bearing Pre-Load

Each axis uses a double-nut, pre-loaded ball screw. The pre-load 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 inches. Ball nut pre-load 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 they are operator-adjustable. Under typical use, the bearings should be adjusted so that observable lost motion is between 0.0012 inches and 0.0018 inches. For reference, the following image 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.

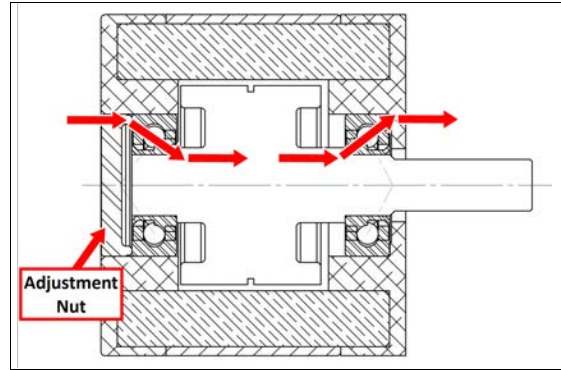


Figure 11-4: Stepper motor cross section.

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 tightened toward the bearing pair, the pre-load increases.

Over time, it may become necessary to adjust the ball screw bearing pre-load to account for bearing wear. Improper ball screw bearing pre-load results in the following issues:

- **If the Ball Screw Bearing Pre-Load is Too Loose**
Excessive backlash in the machine
- **If the Ball Screw Bearing Pre-Load is Too Tight**
Rapid wear and excessive friction

If your ball screw, ball nut, or angular contact bearings are worn, or if your gibs are adjusted too tight, you won't achieve appropriate lost motion values.

11.4.6 Adjust the Geometry 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 use, no adjustment of pinned connections should be necessary. In the event of a hard crash, you can use shims to make minor alignment adjustments between pinned components. Small adjustments (less than 0.010 inches) 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.

11.4.7 Adjust the Spindle Bearings

During operation, sustained high spindle speed will bring the spindle bearings to about 155°F (68°C). This is a normal condition. Spindle bearing pre-load is set at the factory and is not operator-adjustable.

NOTICE! Do not attempt to adjust spindle bearing pre-load. If you do, it could adversely impact spindle balance.

11: MACHINE MAINTENANCE

11.5 Transporting the Machine

11.5 TRANSPORTING THE MACHINE

Before you transport the machine to a new location, you must prepare it to protect against damage.

To prepare the machine for transportation:

1. Place a solid block of wood on the machine table underneath the spindle, and lower the Z-axis so that the wood is slightly compressed.
2. Apply a light mist of WD-40® (or similar water displacement oil) to all exposed, bare metal surfaces to protect against corrosion.
3. Remove the machine from its stand, and secure it to a shipping pallet for vehicle transportation.

NOTICE! You must remove the machine from the stand for transportation. The stand isn't designed to support the weight of the machine during transportation.

Warranty is void if the mill is disassembled. We recognize that there are situations when you need to disassemble the machine, and we've made provisions in the design of the machine to facilitate this. Nevertheless, Tormach cannot be held responsible for alignment, precision, and operating functions after the machine has been disassembled. Test your machine before disassembling it.

The major sub-assemblies of spindle head, column, and base are bolted and dowelled together so that the machine can be separated into smaller components to meet very challenging transportation 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. If you need more detailed advice, we can help. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.

DIAGRAMS AND PARTS LISTS

IN THIS SECTION, YOU'LL LEARN:

- About this machine's components.

NOTICE! Only use Tormach-approved parts when making replacements. If you don't replace parts with those listed in this section, you may void your warranty.

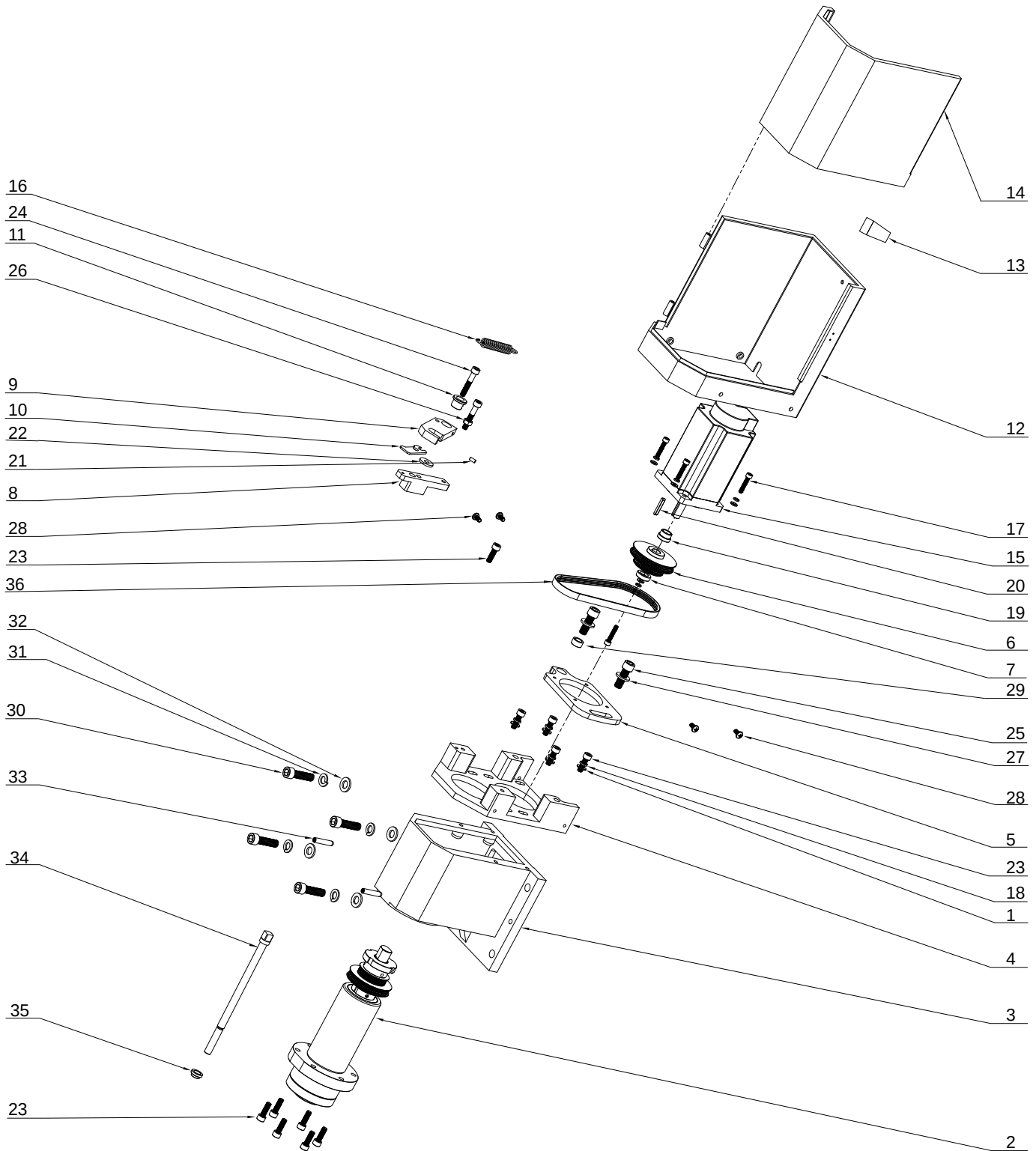
CONTENTS

12.1 Spindle Head	224
12.2 Z-Axis Column	227
12.3 X-Y Axis	229
12.4 Electrical Cabinet	232
12.5 Connections	234
12.6 Stepper Connections	235
12.7 Lubrication/Oil System Diagram	236

12: DIAGRAMS AND PARTS LISTS

12.1 Spindle Head

12.1 SPINDLE HEAD



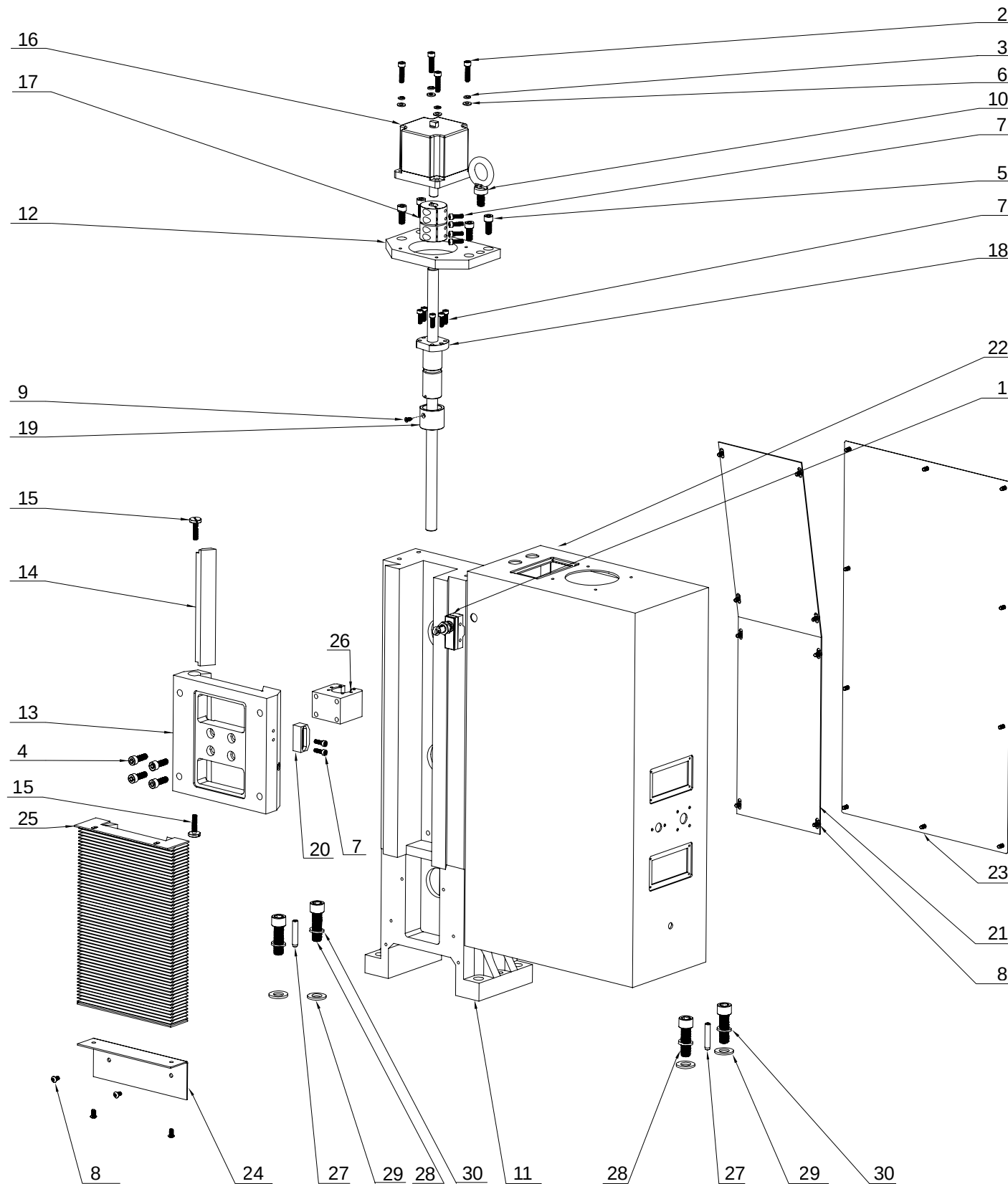
ID	Description
1	M8 Flat Washer (PN 30531)
2	<u>PCNC 440 Spindle Cartridge (PN 33163)</u>
3	Spindle Head (PN 35741)
4	Spindle Motor Mount (PN 35742)
5	Motor Pivot Plate (PN 35743)
6	Spindle Motor Pulley (PN 35744)
7	Pulley Washer (PN 35745)
8	Spindle Lock Block (PN 35746)
9	<u>Spindle Lock, PCNC 440 (PN 35747)</u>
10	Door Jamb (PN 35748)
11	Jamb Bushing (PN 35749)
12	Spindle Motor Cover (PN 35750)
13	Spindle Cover Latch (PN 37312)
14	Motor Cover Door (PN 35751)
15	Spindle Motor (PN 35422)
16	Spindle Lock Spring (PN 35753)
17	Screw, Socket Head Cap, M6 × 30 mm (PN 35754)
18	Lock Washer (PN 35755)
19	Spindle Motor Pulley Bushing (PN 35756)
20	Spindle Motor Key (PN 35757)
21	Dowel, M5 × 10 mm (PN 35758)
22	Plastic Washer (PN 35759)
23	Screw, Socket Head Cap, M8 × 25 mm (PN 35760)
24	Screw, Socket Head Cap, M8 × 45 mm (PN 35761)
25	Screw, Socket Head Cap, M12 × 1.75 × 35 (PN 35762)
26	Nut, Hex, M8 (PN 35763)
27	Washer, Flat, M12 (PN 35764)
28	Screw, Phillips Head Machine, M6 × 12 mm (PN 35765)
29	Spindle Motor Plate Bushing (PN 35766)
30	Screw, Socket Head Cap, M12 × 45 mm (PN 35801)
31	Washer, Split Lock, M12 (PN 35802)
32	Washer, Flat, M12 (PN 35803)

12: DIAGRAMS AND PARTS LISTS

12.1 Spindle Head

ID	Description
33	Pull Dowel, M8 × 40 mm (PN 35804)
34	<u>Drawbar for PCNC 440 (PN 33014)</u>
35	Drawbar Bushing (PN 33168)
36	<u>Belt: PCNC 440 Spindle (PN 35799)</u>

12.2 Z-AXIS COLUMN

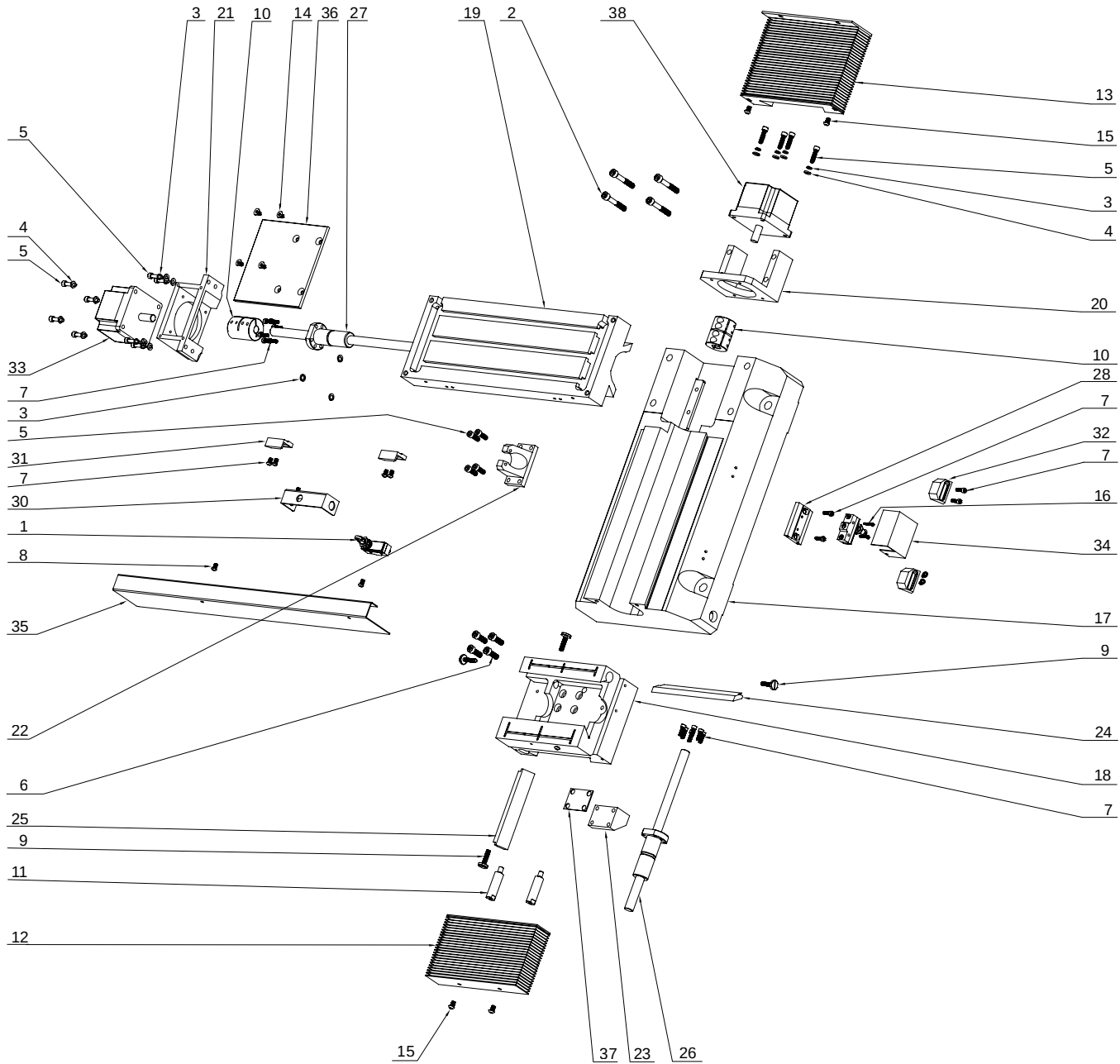


12: DIAGRAMS AND PARTS LISTS

12.2 Z-Axis Column

ID	Description
1	<u>X-/Z-Axis Limit Switch (PN 31860)</u>
2	Screw, Socket Head Cap, M6 × 1 - 25 (PN 31685)
3	Lock Washer (PN 35755)
4	Screw, Socket Head Cap, M8 × 25 mm (PN 35760)
5	Screw, Socket Head Cap, M8 × 20 mm (PN 35771)
6	Washer, Flat, M6 (PN 35772)
7	Screw, Socket Head Cap, M5 × 16 mm (PN 35773)
8	Screw, Button Head Cap, M5 × 0.8 - 10 (PN 35774)
9	Screw, Flat Head Cap, M3 × 6 mm (PN 35775)
10	Eyebolt, M10 (PN 35776)
11	Z-column (PN 35777)
12	Z-motor Plate (PN 35778)
13	Z-saddle (PN 35779)
14	Z-gib (PN 35780)
15	<u>Gib Screw, PCNC 440 (PN 35781)</u>
16	Z-Axis Stepper Motor Assembly, PCNC 440 (PN 35782)
17	Z-coupling 0.5 in.-16 mm (PN 35783)
18	Z-axis 16 mm Ballscrew and Nut (PN 35784)
19	Z-ballscrew Bumper (PN 35785)
20	Z-switch Dog (PN 35786)
21	Z-column Cover (PN 35787)
22	Electrical Cabinet (PN 35788)
23	Electrical Cabinet Cover (PN 35789)
24	<u>Z-Bellows Bracket (PN 35790)</u>
25	<u>Z-Axis Bellows Cover, PCNC 440 (PN 35791)</u>
26	Z-ballnut Bracket (PN 35792)
27	Pull Dowel, M8 × 40 mm (PN 35804)
28	Screw, Socket Head Cap, M14 × 45 mm (PN 35805)
29	Washer, Flat, M14 (PN 35806)
30	Washer, Split Lock, M14 (PN 35807)

12.3 X-Y AXIS



12: DIAGRAMS AND PARTS LISTS

12.3 X-Y Axis

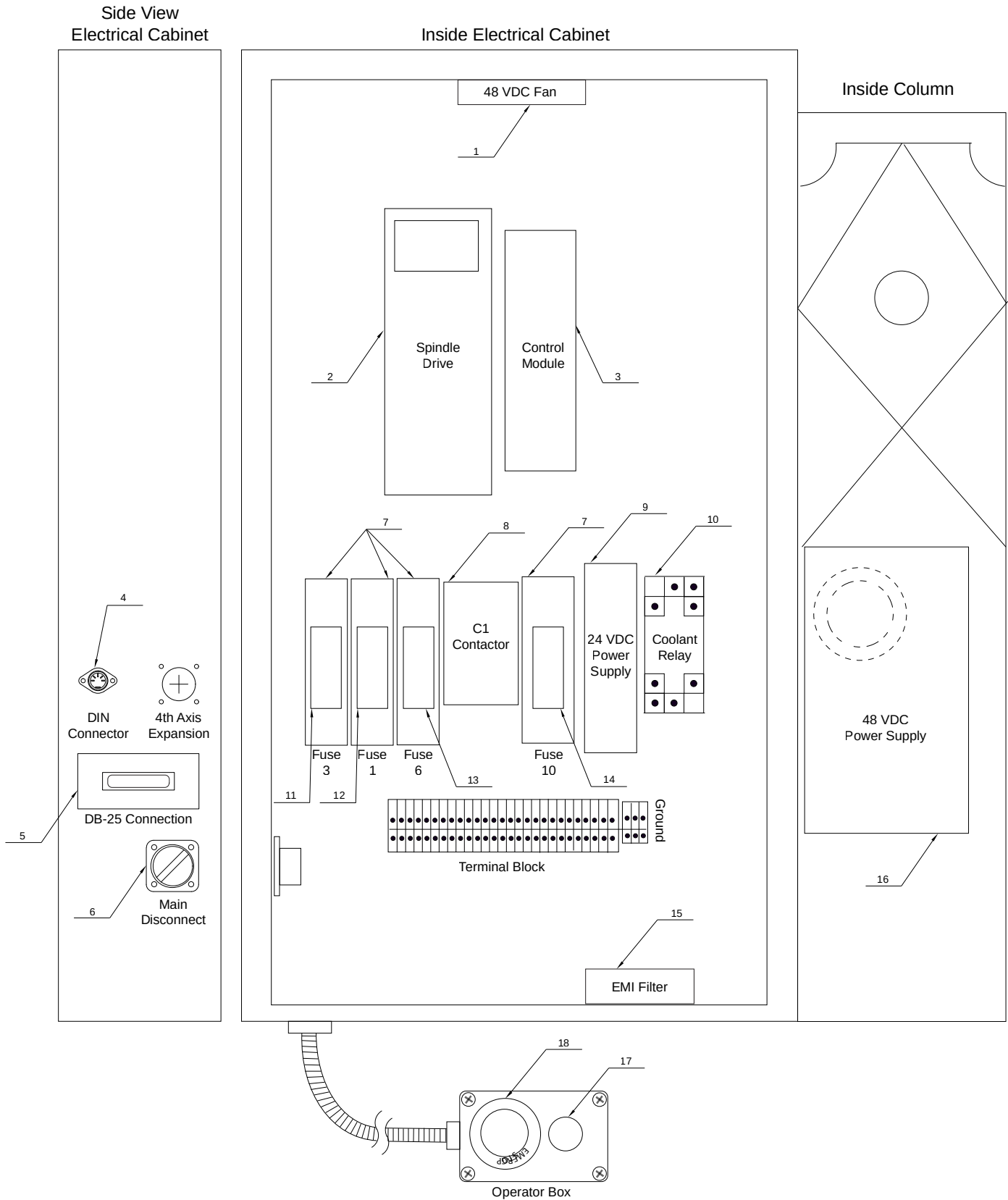
ID	Description
1	<u>X-/Z-Axis Limit Switch (PN 31860)</u>
2	Screw, Socket Head Cap, M8 × 55 mm (PN 30970)
3	Washer, Split Lock, M6 (PN 31379)
4	Washer, Flat, M6 (PN 31513)
5	Screw, Socket Head Cap, M6 × 1 - 25 (PN 31685)
6	Screw, Socket Head Cap, M8 × 25 mm (PN 35760)
7	Screw, Socket Head Cap, M5 × 16 mm (PN 35773)
8	Screw, Button Head Cap, M5 × 10 mm (PN 35774)
9	<u>Gib Screw, PCNC 440 (PN 35781)</u>
10	Coupling 0.5 in.-16 mm (PN 35783)
11	<u>Bellows Stand Off (PN 35793)</u>
12	<u>Y-Axis Front Bellows Cover, PCNC 440 (PN 35794)</u>
13	<u>Y-Axis Rear Bellows Cover, PCNC 440 (PN 35795)</u>
14	Screw, Flat Head, M6 × 10 mm (PN 35796)
15	Screw, Button Head Cap, M6 × 10 mm (PN 35797)
16	Screw, Pan Head Machine, M3 × 20 mm (PN 35798)
17	Machine Base (PN 35808)
18	X-Y Saddle (PN 35809)
19	Machine Table (PN 35810)
20	Y-motor Mount (PN 35811)
21	X-motor Mount (PN 35812)
22	X-ballnut Bracket (PN 35813)
23	Y-ballnut Bracket (PN 35814)
24	X-gib (PN 35815)
25	Y-gib (PN 35816)
26	Y-axis 16 mm Ballscrew and Nut (PN 35818)
27	X-axis 16 mm Ballscrew and Nut (PN 35817)
28	Y-switch Plate (PN 35819)
29	Y-switch Bracket (PN 35820)
30	X-switch Bracket (PN 35821)
31	X-switch Dog (PN 35822)
32	Y-switch Dog (PN 35823)

ID	Description
33	X-Axis Stepper Motor Assembly, PCNC 440 (PN 52930)
34	Y-switch Cover (PN 35825)
35	Front Drip Guard (PN 35826)
36	Table Tray (PN 35827)
37	Y-ballnut Shim (PN 35828)
38	Y-Axis Stepper Motor Assembly, PCNC 440 (PN 52929)

12: DIAGRAMS AND PARTS LISTS

12.4 Electrical Cabinet

12.4 ELECTRICAL CABINET

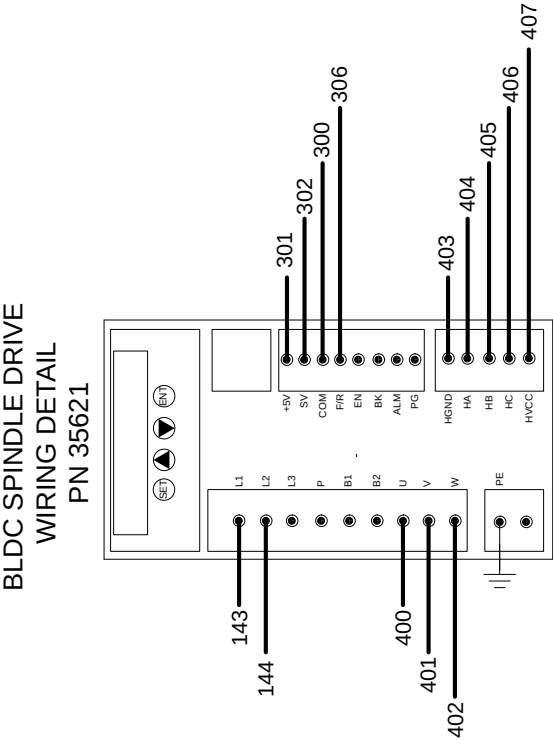


ID	Description
1	48 Vdc Fan (PN 35629)
2	Spindle Drive (PN 35621)
3	PCNC 440 Axis Motor Drive - MX3660 (PN 33374)
4	DIN Connector Assembly (PN 37282)
5	DB-25 Ribbon Cable Assembly (PN 35721)
6	Main Disconnect (PN 35720)
7	<u>Single Fuse Block (PN 30511)</u>
8	<u>C1 Contactor (PN 31659)</u>
9	24 Vdc Power Supply (PN 37153)
10	Coolant Relay (PN 37154)
11	Fuse 3 – 2A (PN 35627)
12	<u>Fuse, 6 A, 500 V, 10 × 38, Glass (PN 30456)</u>
13	<u>Fuse, 15 A, 380 V, 10 × 38, Glass (PN 30455)</u>
14	<u>Fuse, 6 A, 500 V, 10 × 38, Glass (PN 30456)</u>
15	<u>EMI Filter (PN 31440)</u>
16	48 Vdc Power Supply (PN 35628)
17	Push Button (PN 33219)
18	Emergency Stop Button (PN 35980)
19	Emergency Stop Box (PN 39536)
—	<u>Flex Conduit, 16 mm (PN 30627)¹</u>
—	<u>12 mm Flex Conduit (PN 30722)²</u>
—	<u>Flex Conduit Fitting, 13 mm ID × 16 mm OD (PN 30628)</u>
—	<u>Connector for 12 mm Flex Conduit (PN 30723)</u>
¹ 16 mm flex conduit used for: X-axis motor, Y-axis motor, Z-axis motor, spindle motor, emergency stop, and X-Y limit switch. ² 12 mm flex conduit used for Y-axis limit switch.	

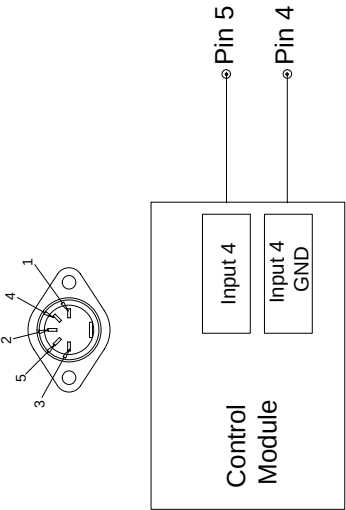
12: DIAGRAMS AND PARTS LISTS

12.5 Connections

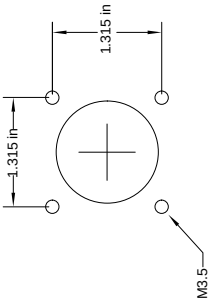
12.5 CONNECTIONS



PN 37282 DIN CONNECTOR ASSEMBLY
Solder side of panel mount
DIN connector on electrical cabinet

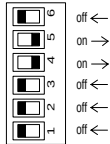


4TH AXIS EXPANSION
ON SIDE OF CABINET



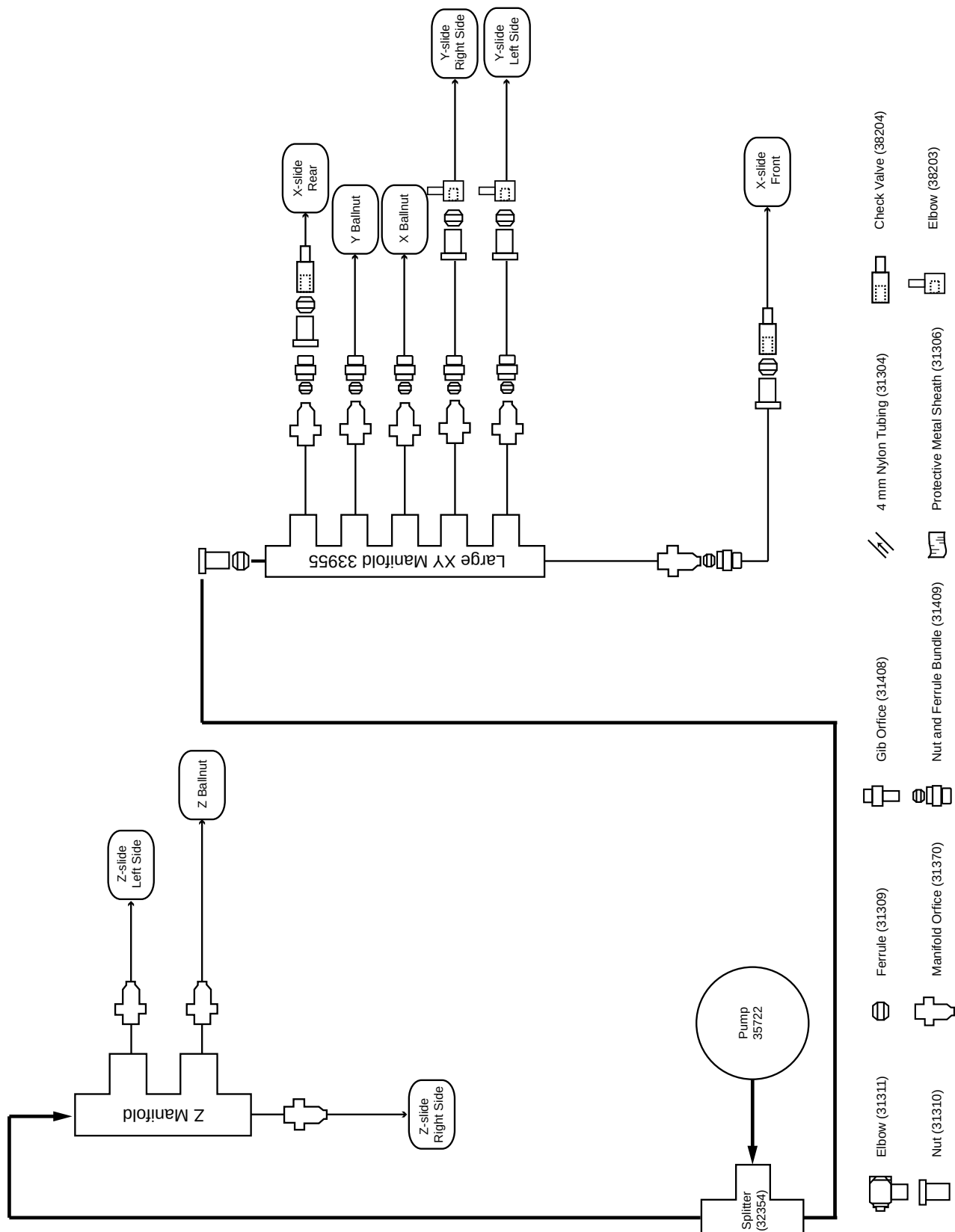
WIRE NUMBER	COLOR	SIZE (AWG)
	GREEN/YELLOW	14 & 16
100	WHITE	14 & 16
102 - 107	RED	16
120 - 126	BLACK	16
140	BLACK	14
142	BLACK	14
143	BLACK	14
144	WHITE	14
161 - 162	BLACK	14
163 - 170	BLUE	16
326 - 327	BLUE	16
400	YELLOW	16
401	RED	16
402	BLACK	16
403	BLACK	26
404	BLUE	26
405	GREEN	26
406	WHITE	26
407	RED	26

12.6 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			
			

12.7 Lubrication/Oil System Diagram

12.7 LUBRICATION/OIL SYSTEM DIAGRAM



ELECTRICAL SCHEMATICS

IN THIS SECTION, YOU'LL LEARN:

- About the electrical schematics for this machine's electronics.

CONTENTS

13.1 Electrical Schematic	238
---------------------------------	-----

13: ELECTRICAL SCHEMATICS

13.1 Electrical Schematic

13.1 ELECTRICAL SCHEMATIC

