

TORMACH® 8L



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®, 8L®, 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. 2026

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.

1: IMPORTANT INFORMATION: PLEASE READ FIRST

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 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" (below).
 - 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)

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.

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

IMPORTANT INFORMATION: PLEASE READ FIRST	3
SAFETY	17
2.1 Intended Use	18
2.2 Machine Standards	19
2.2.1 American National Safety Institute (ANSI).....	19
2.2.2 Occupational Safety and Health Administration (OSHA).....	19
2.3 Safety Overview	20
2.3.1 Safety Messages.....	20
<i>Personal Injury</i>	20
<i>Property Damage</i>	20
2.3.2 Safety Decals.....	20
<i>On the Electrical Cabinet Door</i>	20
<i>Next to the Main Disconnect</i>	21
<i>On the Rear of the Headstock</i>	21
2.3.3 Information Decals.....	22
<i>Serial Number Plate</i>	22
2.4 Machine Safety	23
2.4.1 General Shop Safety.....	23
2.4.2 Operational Safety.....	23
<i>General</i>	23
<i>Tooling</i>	24
<i>Workholding</i>	24
2.4.3 Electrical Safety.....	24
ABOUT YOUR MACHINE	27
3.1 Specifications	28
SITE REQUIREMENTS	29
4.1 General Site and Space Requirements	30
4.1.1 Site Requirements.....	30
4.1.2 Space Requirements.....	30
4.2 Electrical and Power Requirements	31
4.2.1 Electrical Requirements.....	31
4.2.2 Power Requirements.....	31
4.2.3 Options for Non-Conforming Sites.....	31
4.3 Air Requirements	32
INSTALLATION	33
5.1 Before You Begin	34

TABLE OF CONTENTS

5.2 Installation Tools and Items	35
5.3 Move the Pallet	36
5.4 Unpack the Machine Crate	37
5.5 Assemble the Machine Stand	38
5.6 Install Core Components	40
5.6.1 Lift and Move the Machine	40
<i>Secure the Machine to the Stand</i>	40
<i>Secure the Machine to a Workbench</i>	41
5.6.2 Install the Controller Arm	42
5.6.3 Install the Controller	43
<i>Install the PathPilot Operator Console</i>	43
<i>Install the Monitor</i>	46
<i>Install the PathPilot Controller</i>	47
5.7 Verify the Installation	49
5.7.1 Power on the Machine	49
5.7.2 Verify Limit Switch Function	50
5.7.3 Verify Axes Function	50
5.7.4 Verify Spindle Function	51
5.7.5 Power off the Machine	52
5.8 Install Accessory Components	53
5.8.1 Install the Machine Stand Coolant Kit (PN 50931)	53
<i>Adjust the Coolant Pump Motor Strapping</i>	53
5.8.2 Set Up the Quick Change Tool Post Kit	53
<i>Required Tools</i>	54
<i>Install the Tool into the Quick-Change Tool Holder</i>	54
To Use a Boring Bar	54
<i>Align the Tool</i>	55
5.9 Set Up the PathPilot Controller	56
5.9.1 Specify the Date and Time	56
5.9.2 Specify the Keyboard Language	56
5.9.3 Configure the Optional Touch Screen Kit	56
5.9.4 Update PathPilot	56
<hr/> SYSTEM BASICS	<hr/> 57
6.1 System Reference	58
6.1.1 Spindle	58
6.1.2 Axes	58
6.2 Basic Controls Reference	59
6.2.1 Machine Controls	59
6.2.2 PathPilot Interface	59
<hr/> PATHPILOT INTERFACE OVERVIEW	<hr/> 61
7.1 About PathPilot	62
7.2 Notebook Section	63
7.2.1 Main Tab	63

7.2.2 File Tab	63
7.2.3 Settings Tab	63
7.2.4 Offsets Tab	64
7.2.5 Conversational Tab	64
7.2.6 Status Tab	64
7.3 Persistent Controls	65
7.3.1 Program Control Area	65
7.3.2 Position Status Area	65
7.3.3 Manual Control Area	65
7.4 Keyboard Shortcuts	67
7.5 Manage PathPilot Versions	68
7.5.1 Download and Install an Update File from the Controller	68
7.5.2 Install an Update File from a USB Drive	68
7.5.3 Install a Previous Version of an Update File	69
<hr/>	
PATHPILOT TOOLS AND FEATURES	71
<hr/>	
8.1 Create and Load G-Code Files	72
8.1.1 Load G-Code	72
<i>Transfer Files to and From the Controller</i>	<i>72</i>
<i>Preview G-Code Files</i>	<i>72</i>
<i>Access Recent G-Code Files</i>	<i>73</i>
<i>Close the Current Program</i>	<i>73</i>
8.1.2 Edit G-Code	73
<i>Edit G-Code with a Text Editor</i>	<i>73</i>
<i>Edit G-Code with Conversational Programming</i>	<i>73</i>
Change the Step Order	74
Create a New Job Assignment	74
Load an Existing G-Code File	74
Edit a Job Assignment	74
8.1.3 Read G-Code	74
<i>Expand the G-Code Tab</i>	<i>74</i>
About the G-Code Tab	74
<i>Search in the Code</i>	<i>75</i>
<i>Set a New Start Line</i>	<i>75</i>
Lead-In Moves	76
<i>Change the View of the Tool Path Display</i>	<i>76</i>
About the Tool Path Display	77
8.1.4 Use Conversational Programming	77
<i>About Conversational Programming</i>	<i>77</i>
<i>Create an Outside Diameter</i>	<i>77</i>
About OD Turning	79
<i>OD Turning in PathPilot</i>	<i>79</i>
<i>Create an Internal Diameter</i>	<i>79</i>
Basic Internal Diameters	79
Extended Internal Diameters	80

TABLE OF CONTENTS

About ID Turning	81
<i>ID Turning in PathPilot</i>	81
Create a Profile on a Part	82
X-Axis Differences Between Lathes	82
Describe the Stock	82
Identify the Profile Points	83
Identify the Tool	83
Identify the Roughing and Finishing	84
Describe the Tool Geometry	84
About Profiling	84
Create a Face on a Part	85
About Facing	86
Create a Chamfer or Radius on a Part	86
About Chamfer and Radius	87
Create a Groove or Part a Workpiece	87
Create a Groove on a Part	88
Part a Workpiece from the Stock	89
About Grooving and Parting	89
Create Holes on a Part or Tap a Hole	89
Create a Hole on a Part	89
Create Threads in a Hole	90
About Drilling and Tapping	90
<i>Drilling</i>	90
<i>Tapping</i>	91
Create Threads on a Part	91
About Threading	92
8.2 Machine Settings and Accessories	93
8.2.1 Enable an Internet Connection	93
<i>Enable Automatic Updates</i>	93
8.2.2 Change the Network Name	94
8.2.3 Change the Screen Orientation	94
<i>About Portrait Screen Layout</i>	94
8.2.4 Specify the Spindle Pulley Option	95
8.2.5 Specify the Tool Change Method	96
<i>About Gang Tool Changes</i>	96
<i>About Manual Tool Changes</i>	96
<i>About Mixed Tooling Changes</i>	96
8.2.6 Disable Limit Switches	96
<i>About Limit Switches</i>	96
8.2.7 Limit G30 Moves	97
<i>About G30</i>	97
8.2.8 Enable the On-Screen Keyboard	97
<i>About Soft Keyboards</i>	97
8.2.9 Enable Tooltips	97
8.2.10 Enable the USB M-Code I/O Interface Kit	98

8.2.11 Enable Feeds and Speeds Suggestions in Conversational Routines.....	98
8.2.12 Use a USB Camera.....	98
<i>About USB Cameras.....</i>	<i>98</i>
<i>Manual Recording.....</i>	<i>99</i>
<i>Automatic E-Stop Loop Recording ("Dashcam").....</i>	<i>99</i>
<i>Review Video and Image Files.....</i>	<i>100</i>
File Naming Convention.....	100
<i>G-Code Commands.....</i>	<i>100</i>
File Naming Conventions.....	100
Use M01 to Take Pictures.....	100
8.3 Set Up G-Code Programs.....	102
8.3.1 Set Tool Geometry Offsets.....	102
8.3.2 Touch Off the Tool Geometry Offsets.....	102
<i>Touch X.....</i>	<i>102</i>
<i>Touch Z.....</i>	<i>103</i>
<i>About Tool Offsets.....</i>	<i>103</i>
8.3.3 Set Work Offsets.....	103
<i>About Work Offsets.....</i>	<i>103</i>
8.3.4 View Work Offsets.....	104
8.3.5 View Available G-Code Modes.....	104
8.4 Run G-Code Programs.....	105
8.4.1 Bring the Machine Out of Reset.....	105
<i>About Reset Mode.....</i>	<i>105</i>
8.4.2 View the Active Axis to Jog.....	105
8.4.3 Jog the Machine.....	105
<i>Jog in Continuous Velocity Mode.....</i>	<i>106</i>
About Continuous Velocity Jogging.....	106
<i>Jog in Step Mode.....</i>	<i>106</i>
About Step Jogging.....	106
8.4.4 View the Current Machine Position.....	106
8.4.5 Reference the Machine.....	106
<i>About Referencing.....</i>	<i>107</i>
8.4.6 Start a Program.....	107
<i>About Cycle Start.....</i>	<i>107</i>
<i>Cycle Start Reference.....</i>	<i>107</i>
8.4.7 Stop Machine Motion.....	107
8.4.8 Operate the Coolant Pump.....	107
<i>About Coolant.....</i>	<i>108</i>
8.4.9 View the Active G-Code Modes.....	108
8.4.10 View the Distance to Go.....	108
<i>About Distance to Go.....</i>	<i>108</i>
8.5 Control G-Code Programs.....	109
8.5.1 Use the Feed Hold Function.....	109
<i>About Feed Hold.....</i>	<i>109</i>
8.5.2 Use the Feed Rate Override Function.....	109

TABLE OF CONTENTS

<i>About Feed Rate Override</i>	109
8.5.3 Use M01 Break Mode.....	110
<i>About M01 Break</i>	110
8.5.4 Use the Maxvel Override Function.....	110
<i>About Maxvel Override</i>	110
8.5.5 Use Single Block Mode.....	110
<i>About Single Block</i>	110
8.5.6 Use the Spindle Override Function.....	110
<i>About Spindle Override</i>	111
8.5.7 Change the Feed Rate.....	111
<i>About Feed Rates</i>	111
8.5.8 Change the Spindle Speed.....	111
<i>About Spindle Controls</i>	112
8.5.9 Change the Feed Rate Mode.....	112
<i>About Feed Rate Modes</i>	112
Units per Minute Mode (G94).....	113
Units per Revolution Mode (G95).....	113
8.5.10 Change the Spindle Speed Command Mode.....	113
<i>About Spindle Speed Command Modes</i>	113
Constant Surface Speed Mode (G96).....	113
Constant RPM Mode (G97).....	113
<i>About Spindle Controls</i>	113
8.5.11 Change the Tool Number.....	114
8.5.12 Use a G30 Position.....	114
<i>About G30</i>	115
8.5.13 Manually Enter Commands.....	115
<i>About the MDI Line DRO Field</i>	115
<i>Admin Commands Reference</i>	115
8.5.14 Copy Recently Entered Commands.....	116
8.5.15 Use Feeds and Speeds Suggestions.....	116
<i>Refresh DRO Field Values</i>	117
<i>Enable Feeds and Speeds Suggestions in Conversational Routines</i>	117
<i>Create Tool Descriptions</i>	117
Manually Enter Tool Descriptions.....	117
Automatically Generate Tool Descriptions.....	118
<i>Tool Keywords Reference</i>	118
8.5.16 Use Cycle Counters (M30 and M99).....	119
<i>Monitor Cycle Counters</i>	120
<i>Change Cycle Counter Values</i>	120
8.6 System File Management	121
8.6.1 Manage System Files.....	121
8.6.2 Create Backup Files.....	121
<i>About Backup Files</i>	122
8.6.3 Restore Backup Files.....	122
8.6.4 Import and Export the Tool Table.....	123

<i>Import a .csv File</i>	123
<i>Export the Tool Table as a .csv File</i>	123
BASIC OPERATIONS	125
9.1 Start the Machine	126
9.2 Bring the Machine Out of Reset	127
9.2.1 About Reset Mode.....	127
9.3 Reference the Machine	128
9.3.1 About Referencing.....	128
9.4 Jog the Machine	129
9.4.1 About Jogging.....	129
9.4.2 About Continuous Velocity Jogging.....	129
9.4.3 About Step Jogging.....	129
9.4.4 Jog Controls Reference.....	130
<i>Jogging in PathPilot</i>	130
<i>Jogging with the Keyboard</i>	130
<i>Jogging with the (Optional) Jog Shuttle or Operator Console Pendant</i>	130
9.5 Change the Spindle Speed Range	131
9.6 Load Material into the Spindle	132
9.7 Load G-Code	133
9.7.1 Transfer Files to and From the Controller.....	133
9.8 Set Work Offsets	134
9.8.1 About Work Offsets.....	134
9.9 Set Tool Geometry Offsets	135
9.10 Touch Off the Tool Geometry Offsets	136
9.10.1 Touch X.....	136
9.10.2 Touch Z.....	136
9.10.3 About Tool Offsets.....	136
9.10.4 Import and Export the Tool Table.....	137
<i>Import a .csv File</i>	137
<i>Export the Tool Table as a .csv File</i>	137
9.11 Operate the Coolant Pump	138
9.11.1 About Coolant.....	138
PROGRAMMING	139
10.1 Before You Begin	140
10.1.1 Definitions.....	140
<i>Linear Axes</i>	140
<i>Controlled Point</i>	140
<i>Coordinated Linear Motion</i>	141
<i>Feed Rate</i>	141
<i>Arc Motion</i>	141
<i>Dwell</i>	141
<i>Units</i>	141
<i>Current Position</i>	141

TABLE OF CONTENTS

<i>Work Offsets</i>	141
<i>Selected Plane</i>	141
<i>Tool Table</i>	141
<i>Wear Offsets</i>	141
<i>Path Control Modes</i>	142
<i>Feed and Speed Override Controls</i>	142
<i>Block Delete Control</i>	142
<i>Optional Program Stop Control</i>	142
10.2 Programming Overview	143
10.2.1 About G-Code Programming Language.....	143
10.2.2 G-Code Formatting Reference.....	143
<i>Line Numbers</i>	143
<i>Words</i>	143
<i>Letters</i>	144
<i>Values</i>	144
<i>Order of Execution</i>	145
<i>Modal Groups</i>	146
<i>Comments</i>	147
10.2.3 Supported G-Codes Reference.....	147
10.3 Programming G-Code	148
10.3.1 About the Examples Used.....	148
10.3.2 Rapid Linear Motion (G00).....	148
10.3.3 Linear Motion at Feed Rate (G01).....	148
10.3.4 Arc at Feed Rate (G02 and G03).....	149
<i>Radius Format Arc</i>	149
<i>Center Format Arc</i>	149
10.3.5 Dwell (G04).....	149
10.3.6 Lathe Diameter Mode (G07).....	149
10.3.7 Set Offsets (G10).....	149
<i>Set Tool Table (G10 L1)</i>	150
<i>Set Coordinate System (G10 L2)</i>	150
<i>Set Tool Table (G10 L10)</i>	150
<i>Set Tool Table (G10 L11)</i>	150
<i>Set Coordinate System (G10 L20)</i>	151
10.3.8 Plane Selection (G17, G18, G19).....	151
10.3.9 Length Units (G20 and G21).....	151
10.3.10 Return to Predefined Position (G28 and G28.1).....	151
10.3.11 Return to Predefined Position (G30 and G30.1).....	151
10.3.12 Spindle-Synchronized Motion (G33).....	152
10.3.13 Rigid Tapping (G33.1).....	152
10.3.14 Cutter Compensation (G40, G41, G42).....	153
10.3.15 Dynamic Cutter Compensation (G41.1 and G42.1).....	153
10.3.16 Absolute Coordinates (G53).....	153
10.3.17 Select Work Offset Coordinate System (G54 to G54.1 P500).....	153
10.3.18 Set Exact Path Control Mode (G61).....	154

10.3.19 Set Blended Path Control Mode (G64).....	154
10.3.20 Threading Cycle (G76).....	154
<i>Optional Settings</i>	154
10.3.21 Tapping Cycle (G84).....	155
10.3.22 Distance Mode (G90 and G91).....	156
10.3.23 Arc Distance Mode (G90.1 and G91.1).....	156
10.3.24 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	156
10.3.25 Feed Rate Mode (G93, G94, and G95).....	156
10.3.26 Spindle Control Mode (G96 and G97).....	157
10.4 Programming M-Code	158
10.4.1 Supported M-Codes Reference.....	158
10.4.2 Program Stop and Program End (M00, M01, M02, and M30).....	158
10.4.3 Display Information and Capture Images During an M00 or M01 Break.....	159
<i>Display Information with Images</i>	159
<i>Display Information with Text</i>	159
<i>Capture Images with a USB Camera</i>	159
10.4.4 Spindle Control (M03, M04, and M05).....	159
10.4.5 Coolant Control (and M09).....	159
10.4.6 Automatic Collet Closer Control (M10 and M11).....	159
10.4.7 Override Control (M48 and M49).....	160
10.4.8 Feed Override Control (M50).....	160
10.4.9 Spindle Speed Override Control (M51).....	160
10.4.10 Set Current Tool Number (M61).....	160
10.4.11 Call Subroutine (M98).....	160
10.4.12 Return from Subroutine (M99).....	160
10.4.13 USB Camera Control (M301, M302, M303).....	160
10.5 Programming Input Codes	161
10.5.1 Feed Rate (F).....	161
10.5.2 Spindle Speed (S).....	161
10.5.3 Change Tool Number (T).....	161
10.6 Advanced Programming	162
10.6.1 Parameters.....	162
<i>Parameters Reference</i>	162
Parameter Syntax.....	162
Parameter Scope.....	162
Behavior of Uninitialized Parameters.....	162
Parameter Mode.....	162
Persistence and Volatility.....	162
Intended Use.....	162
<i>Numbered Parameters Reference</i>	163
Read-Only Parameters.....	163
<i>Subroutine Parameters Reference</i>	163
<i>Named Parameters Reference</i>	163
10.6.2 Expressions.....	164
<i>Binary Operators Reference</i>	164

TABLE OF CONTENTS

<i>Functions Reference</i>	164
10.6.3 Subroutines.....	165
<i>Subroutines Reference</i>	165
<i>Conditional Subroutines Reference</i>	166
if/endif.....	166
if/elseif/else/endif.....	166
<i>Repeating Subroutines Reference</i>	166
<i>Looping Subroutines Reference</i>	166
do/while.....	167
while/endwhile.....	167
MACHINE MAINTENANCE	169
11.1 Maintenance Safety	170
11.1.1 All Maintenance Procedures.....	170
11.1.2 Swarf Maintenance Procedures.....	170
11.1.3 Coolant Maintenance Procedures.....	170
11.2 Maintenance Schedules	171
11.2.1 Daily.....	171
11.2.2 Weekly.....	171
11.2.3 Monthly.....	171
11.2.4 Semi-Annually.....	171
11.2.5 Annually.....	171
11.3 Clean the Coolant System	172
11.3.1 About Cutting Fluid.....	172
11.3.2 Cutting Fluid Reference.....	172
11.4 Examine the Enclosure Windows	174
11.5 Examine the Spindle Belt	175
11.6 Lubricate the Machine	176
11.6.1 About the Manual Oiler.....	176
11.6.2 Lubrication System Reference.....	176
11.7 Prevent Rust	177
11.7.1 Remove Rust.....	177
TROUBLESHOOTING	179
12.1 Troubleshooting Safety	180
12.2 Getting Help	181
12.3 Required Tools	182
12.4 Frequently Found Problems	183
12.5 Power Distribution Subsystem	184
12.5.1 The Controller Won't Power On.....	184
12.5.2 The Coolant Pump Won't Run.....	184
12.6 Control Power Subsystem	186
12.6.1 The Machine Won't Power On.....	186
12.7 Axes Drive Subsystem	188
12.7.1 All Axes Won't Move When Commanded.....	188


12.7.2 One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move.....	189
12.7.3 Axis Movement is Noisy.....	191
12.7.4 Can't Reference All Axes.....	192
12.7.5 Lost Motion on Axis Travel.....	193
12.8 Spindle Drive Subsystem.....	196
12.8.1 Spindle Turns in Reverse.....	196
12.8.2 The Spindle Won't Turn.....	197
<i>Run and Direction Commands Reference.....</i>	<i>199</i>
<i>Spindle VFD Trip Reference.....</i>	<i>200</i>
12.9 Operator Console Troubleshooting.....	201
12.9.1 The Screen Doesn't Respond to Touch Inputs.....	201
12.9.2 The Screen Doesn't Display an Image or Respond to Power Button.....	202
12.9.3 The Screen is Scrambled or Illegible.....	202
12.9.4 The Knobs Don't Respond.....	203
12.9.5 The Buttons Don't Respond.....	204
<hr/>	
DIAGRAMS AND PARTS LISTS.....	205
<hr/>	
13.1 Headstock.....	206
13.2 Stand.....	208
13.3 Tailstock.....	210
13.4 X-Axis.....	212
13.5 Z-Axis.....	215
13.6 Electrical Cabinet.....	217
<hr/>	
ELECTRICAL SCHEMATICS.....	219
<hr/>	
14.1 115 Vac Power (Sheet 2).....	220
14.2 24 Vdc Controls (Sheet 3).....	221
14.3 Axis Drive Bus (Sheet 4).....	222
14.4 Spindle Drive (Sheet 5).....	223
14.5 Machine Control Board (Sheet 6).....	224
14.6 Limit/Door Switches (Sheet 7).....	225
14.7 Accessory Power (Sheet 8).....	226
14.8 Grounds (Sheet 9).....	227
14.9 Grounds Terminal Strip (Sheet 10).....	228
14.10 AC Terminal Strip (Before EMI1) (Sheet 11).....	229
14.11 AC Terminal Strip (After EMI1) (Sheet 12).....	230
14.12 Control Terminal Strip (Sheets 13-15).....	231
14.13 Limit Switches Terminal Strip (Sheet 16).....	234
14.14 Operator Console Schematic.....	235



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

2.1 Intended Use	18
2.2 Machine Standards	19
2.3 Safety Overview	20
2.4 Machine Safety	23

2: SAFETY

2.1 Intended Use

2.1 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) 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
- Wood

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

2.2 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.

2.2.1 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 B11.5-1984 (R1994)** *Lathes*
- **ANSI B11.22-2002** *Safety Requirements for Turning Centers and Automatic Numerically Controlled Turning Machines*

2.2.2 Occupational Safety and Health Administration (OSHA)

- **OSHA 1910.212** *General Requirements for All Machines*

2: SAFETY

2.3 Safety Overview

2.3 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.

2.3.1 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.

2.3.2 Safety Decals

Before operating the machine in any way, you must read and

understand all installed safety decals on the machine and equipment. Do not remove any safety decals. If any safety decals become worn or damaged, contact Tormach Technical Support for guidance on receiving replacement decals.

The following types of safety symbols are on the decals:

- **Warning** ⚠ This symbol indicates a hazard which, if not avoided, can result in personal injury or property damage.
- **Prohibition** ⚡ This symbol indicates an action that shall not be taken or that shall be stopped.
- **Mandatory Action** 🔒 This symbol indicates an action that you must take to avoid a hazard.

On the Electrical Cabinet Door

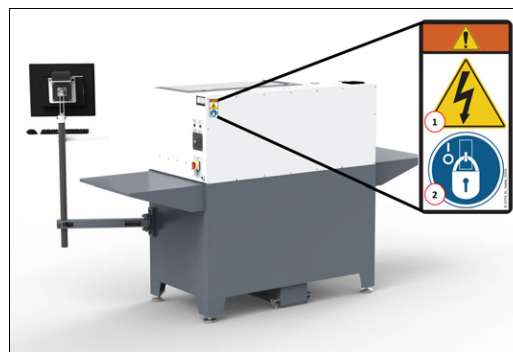


Figure 2-1: Example of a safety decal on the electrical cabinet door.

1. **WARNING! Electrocutation Hazard.** Points in the electrical cabinet contain high voltages, which can electrocute or shock you, causing death or serious injury. Even after the machine is powered off, electronic devices in the electrical cabinet can retain dangerous electrical voltages. Use caution when servicing the machine inside the electrical cabinet.
2. **Lockout/Tagout.** Before servicing the machine, you must power off the machine and use an approved lockout/tagout device to secure the Main Disconnect switch in the **OFF** position. Points in the electrical cabinet contain high voltages, which can electrocute or shock you, causing death or serious injury.

Next to the Main Disconnect

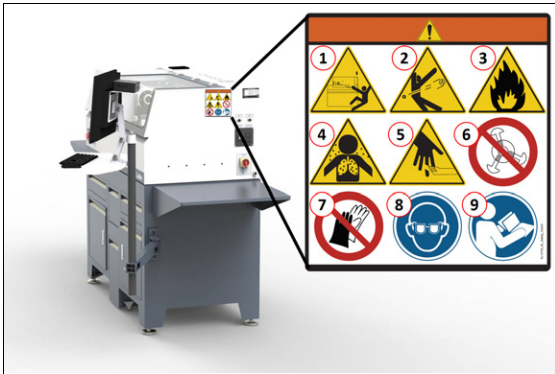


Figure 2-2: Example of a safety decal next to the Main Disconnect.

1. **WARNING! Ejection Hazard.** Fixtures, tooling, workpieces, or other loose items can become dangerous projectiles and can cause death or serious injury. Before operating this machine in any way, you must verify that you have appropriately secured all components.
2. **WARNING! Loose Workpiece or Workholding Hazard.** You must verify that the workpieces are securely clamped into the workholding device, and verify that the workholding device is securely fastened.
3. **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. Before operating the machine in any way, you must read all Safety Data Sheets (SDSs) for any workpiece materials, coatings, coolants, lubricants, and other consumables used.
4. **WARNING! Inhalation Hazard.** The machine and its enclosure do not protect you from airborne particulates. Chips, dust, and vapors from certain materials can be toxic or otherwise harmful. Before operating the machine in any way, you must read all Safety Data Sheets (SDSs) for any workpiece materials, coatings, coolants, lubricants, and other consumables used.
5. **WARNING! Cut Hazard.** Tools and swarf can cut you. Only hold tools by the tool holder. Before inserting or removing tools from the machine, you must verify that all motion is completely stopped.
6. **Don't Use Extended Jaws.** You must not use jaws other than those made specifically for this machine. Extended

jaw can cause the jaws to break or the chuck to fail. If you use extended jaws, there's a risk of property damage, serious injury, or death.

7. **Don't Wear Gloves.** You must not operate this machine while wearing gloves. If you do, the rotating components could draw you into the machine while it's powered on and operating.
8. **Personal Protective Equipment: Eyes.** Prevent injury by always wearing protective safety eyewear. Before operating this machine in any way, you must verify that your eyewear is impact-resistant and rated for ANSI 787+.
9. **Operator Knowledge.** 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.

On the Rear of the Headstock

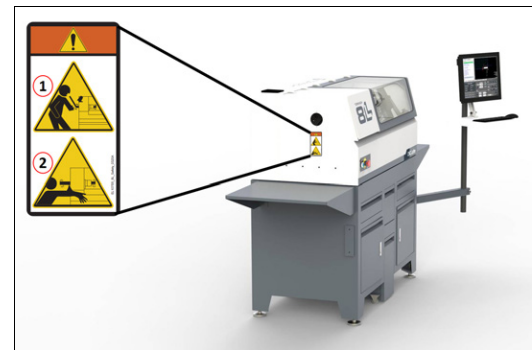


Figure 2-3: Example of safety decals on the rear of the headstock.

1. **WARNING! Bar Whip Hazard.** Unsupported bar stock that extends from the rear of the drawtube can bend during operation, causing it to whip unpredictably. Never extend an unsupported bar stock past the left end of the spindle bore. If you do, there's a risk of serious injury or death.
2. **WARNING! Entanglement / Entrapment Hazard.** The machine operates under automatic control — it can start at any time and crush, cut, entangle, or pinch body parts. Always keep clear of positions on the machine where unexpected or unintended machine motion could cause harm. Before operating this machine in any way, you must verify that all operators know the location of the machine's Emergency Stop button.

2: SAFETY

2.3 Safety Overview

2.3.3 Information Decals

Before operating the machine in any way, you must locate and become familiar with all installed information decals on the machine and equipment.

Serial Number Plate

The serial number plate is on the side of the electrical cabinet, near the Main Disconnect switch.

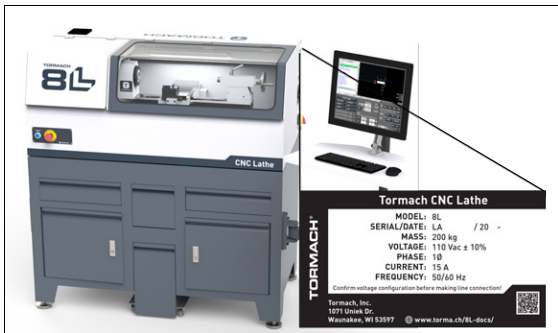


Figure 2-4: Example of the serial number plate on the side of the electrical cabinet.

2.4 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.

2.4.1 General Shop Safety	23
2.4.2 Operational Safety	23
2.4.3 Electrical Safety	24

2.4.1 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.
- ✓ 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.

2.4.2 Operational Safety

General

- ✓ Understand that the machine is automatically controlled and can start at any time.
- ✓ 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 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.
- ✓ Understand that this product can expose you to chemicals which are known to the State of California to cause cancer and birth defects or other reproductive harm. For more information, go to www.P65Warnings.ca.gov.
- ✓ Dispose of scrap and swarf according to local regulations and guidelines.
- ✓ Thoroughly read all safety precautions and instructions.

2: SAFETY

2.4 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.
- ✘ Never enter the machining envelope.
- ✘ Never reach around a guard.
- ✘ Never allow the machine to run with the enclosure door open.
- ✘ 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.
- ✔ 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 extend an unsupported bar stock past the left end of the spindle bore.
- ✘ Never machine any materials without first securing them to the tailstock.
- ✘ Never operate the machine with the tailstock removed.
- ✘ Never leave tools, stock, or other loose items inside the machine.

2.4.3 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 31).

- ✓ 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.



ABOUT YOUR MACHINE

IN THIS SECTION, YOU'LL LEARN:

- About this machine's specifications.

CONTENTS

3.1 Specifications	28
--------------------------	----

3: ABOUT YOUR MACHINE

3.1 Specifications


3.1 SPECIFICATIONS

Travels	
X-Axis	4.5" (114 mm)
Z-Axis	10" (254 mm) with tailstock
Spindle	
Spindle Speed	<ul style="list-style-type: none">• Low 180 rpm to 2500 rpm• High 350 rpm to 5000 rpm
Through Spindle Bore	1" (25 mm)
Maximum Workpiece Length	10" (254 mm) with tailstock
Maximum Feed Rate	
X- and Z-Axis	150 IPM (3.8 m/min)
Tailstock	
Taper	MT2
Diameter	0.9" (22 mm)
Travel	2.2" (55 mm)
Machine Specifications	
Table Length	7.5" (191 mm)
Table Width	4" (102 mm)
Maximum Swing Over Bed	8" (200 mm)
Maximum Swing Over Carriage	4" (100 mm)
Machine Footprint	50" × 26" (1.2 m × 0.7 m)
Overall System Height	52" (1.3 m)
Typical System Weight	838 lb (380 kg)
Coolant Tank Capacity	6 gal (23 liters)
Power	
Primary Power Required	Single-Phase 115 Vac, 50/60 Hz
Recommended Circuit Amperage	Dedicated 15 A breaker

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

4.1 General Site and Space Requirements	30
4.2 Electrical and Power Requirements	31
4.3 Air Requirements	32

4: SITE REQUIREMENTS

4.1 General Site and Space Requirements

4.1 GENERAL SITE AND SPACE REQUIREMENTS

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

4.1.1 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.



Note: Compressed air is only required for certain accessories.

- 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.

4.1.2 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** 50" × 26" (1.2 m × 0.7 m)
- **Machine Height** 52" (1.3 m)

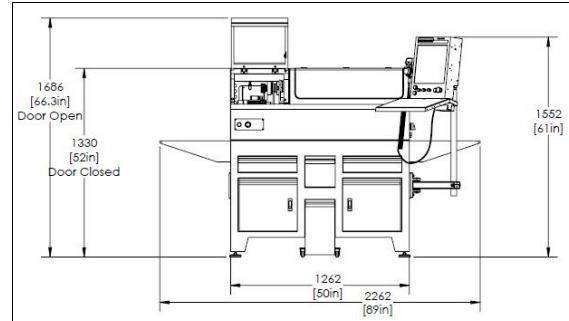


Figure 4-1: Dimensions of the machine itself, as viewed from the front.

4.2 ELECTRICAL AND POWER REQUIREMENTS

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

4.2.1 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

4.2.2 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 variable frequency drive (VFD) spindle controller.
- **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.

4.2.3 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).

4: SITE REQUIREMENTS

4.3 Air Requirements


4.3 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.
- **Dry Air** We recommend using a compressed air dryer, desiccator, or filter between the air compressor and the machine.
- **Lubricated Air** You must lubricate the air with air tool oil.

INSTALLATION

IN THIS SECTION, YOU'LL LEARN:

- About the installation process required for this machine.

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

CONTENTS

5.1 Before You Begin	34
5.2 Installation Tools and Items	35
5.3 Move the Pallet	36
5.4 Unpack the Machine Crate	37
5.5 Assemble the Machine Stand	38
5.6 Install Core Components	40
5.7 Verify the Installation	49
5.8 Install Accessory Components	53
5.9 Set Up the PathPilot Controller	56

5: INSTALLATION

5.1 Before You Begin

5.1 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.

5.2 INSTALLATION TOOLS AND ITEMS

Before uncrating and installing your machine, collect the following tools and items.

- 19 mm socket wrench
- Cardboard piece, large
- Carpenter's level
- Dead-blow hammer (or similar)
- 500-lb capacity engine hoist
- Lifting straps
- Metric hex wrench set
- Pallet jack
- Phillips screwdriver
- Safety eyewear that meets ANSI Z87+
- Snips
- Wood blocks
- Work gloves

5: INSTALLATION

5.3 Move the Pallet

5.3 MOVE THE PALLET

Tools and Items Required

- Pallet jack

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).

5.4 UNPACK THE MACHINE CRATE

Tools and Items Required

- 19 mm socket wrench
- Phillips screwdriver
- Safety eyewear that meets ANSI Z87+
- Snips
- Wood blocks
- Work gloves

⚠ 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. Remove the screws from the bottom of the crate with a Phillips screwdriver.

⚠ 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.

4. Before you lift the top of the shipping crate off of the pallet, verify that you have assistance to help you. We recommend lifting it with at least two people.
5. Remove the top of the shipping crate and set it aside.
6. Remove and discard the plastic wrapped around the machine and stapled to the pallet.
7. 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.
8. Identify the toolbox inside the machine's enclosure. Set

the toolbox aside for now — it includes the machine's four leveling feet, which you'll need later in the installation procedure.

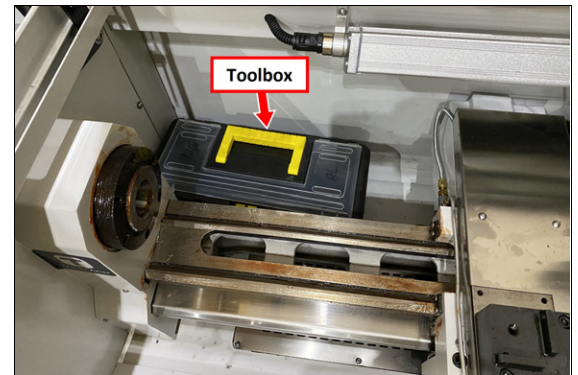


Figure 5-1: Toolbox inside the machine's enclosure.

9. Remove the four bolts from the bottom of the pallet with a 19 mm socket wrench. Discard the bolts.


5: INSTALLATION

5.5 Assemble the Machine Stand


5.5 ASSEMBLE THE MACHINE STAND

Tools and Items Required

- 19 mm socket wrench
- Cardboard piece, large
- Phillips screwdriver
- Safety eyewear that meets ANSI Z87+
- Work gloves


 **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. Remove the screws from the bottom of the crate with a Phillips screwdriver.

 **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.

3. Before you lift the top of the shipping crate off of the pallet, verify that you have assistance to help you. We recommend lifting it with at least two people.
4. Remove the top of the shipping crate and set it aside.
5. Remove the plastic wrap from the machine stand.
6. 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.
7. Remove the coolant tank from the right-hand cabinet of the machine stand, and set it aside.
8. The door for the right-hand cabinet is shipped inside the left-hand cabinet. Remove the plastic wrap from the door and install into the right-hand cabinet opening.

9. Find the accessory box inside the machine's stand, which includes the legs for the machine's feet assembly. Set it aside for later installation.
10. Find the tool box inside the machine's enclosure, which includes the machine's feet, the Ethernet cable, and four M10 lifting eyelets. Remove the machine's feet, and set the tool box aside.
11. Remove the four screws securing the machine stand to the pallet with a 19 mm socket wrench.
12. To install the machine stand's feet, you must first lay it on its front surface. Put a large piece of cardboard (or similar) on the floor to protect the machine stand.

 **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.

13. Lift the machine stand off of the pallet, and put it on the cardboard.
14. Open the accessory box that you set aside in Step 9, and identify the four legs.
15. Thread one leg into each corner of the machine stand.



Note: You can insert a screwdriver into the holes on the side of the legs to tighten them.

16. Find the four feet that you set aside in Step 10.

17. Thread one foot into each leg, making sure that the leg and foot assembly is 8-1/4 in. tall.

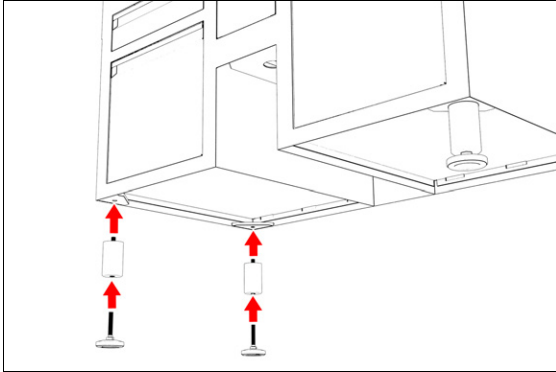


Figure 5-2: Installing the legs and feet onto the machine stand.

⚠ 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.

18. Lift the machine stand off of the floor, and set it down onto its feet.

5: INSTALLATION

5.6 Install Core Components

5.6 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:

5.6.1 Lift and Move the Machine.....	40
5.6.2 Install the Controller Arm.....	42
5.6.3 Install the Controller.....	43

5.6.1 Lift and Move the Machine

Tools and Items Required

- 500-lb capacity engine hoist
- Lifting straps

1. Find the four M10 lifting eyelets provided in the tool box. You'll use them as lifting points to move the machine off of the pallet.
2. Install the two M10 lifting eyelets into each end of the machine by hand in the locations shown in the following image. Verify that the lifting eyelets are installed hand tight.

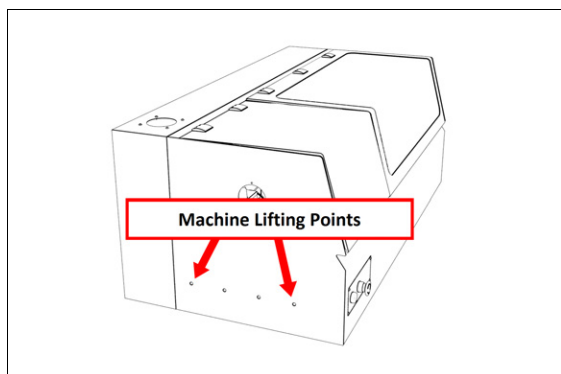


Figure 5-3: Lifting points on the left side of the machine.



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.

3. Using lifting straps (not provided) on a 500-lb capacity engine hoist, lift the machine up from the pallet.

4. Depending on where you're installing the machine, do one of the following:

- **Tormach Machine Stand** Go to "Secure the Machine to the Stand" (below).
- **500-lb Capacity Workbench** Go to "Secure the Machine to a Workbench" (on the next page).

Secure the Machine to the Stand

1. Remove the drip tray from the bottom of the machine.

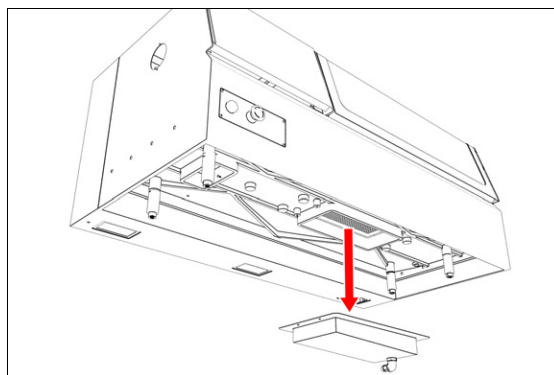


Figure 5-4: Removing the drip tray from the bottom of the machine.

2. Identify the two fixed-length standoffs on the bottom of the machine, and verify that they're tight.
3. Measure the length of the two fixed standoffs. Then, adjust the two adjustable standoffs until they're the same length as the fixed standoffs.
4. Tighten the jam nuts on the two adjustable standoffs with a 19 mm wrench.
5. Identify the four M14 set screws included in the hardware box from the stand.
6. Thread one of the set screws into each of the standoffs on the bottom of the machine. Then, tighten them using a hex key.

- Identify the coolant tube in the chip basket. Then, attach the coolant tube onto the fitting on the bottom of the machine.

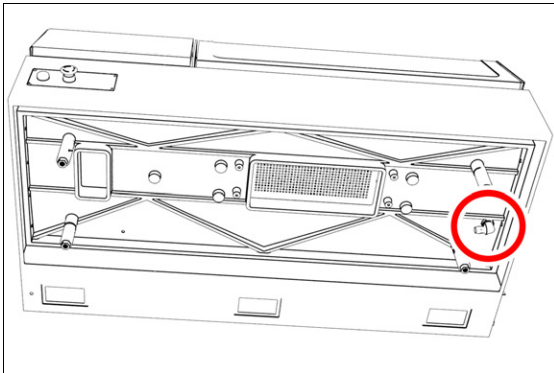


Figure 5-5: Coolant tube fitting on the bottom of the machine.

NOTICE! You must route the coolant tube through the provided hole in the stand, as described in the following step. If you don't route the tube through the stand, it may result in coolant leaks.

- Carefully lower the machine onto the machine stand while routing the coolant tube through the hole in the middle of the stand.

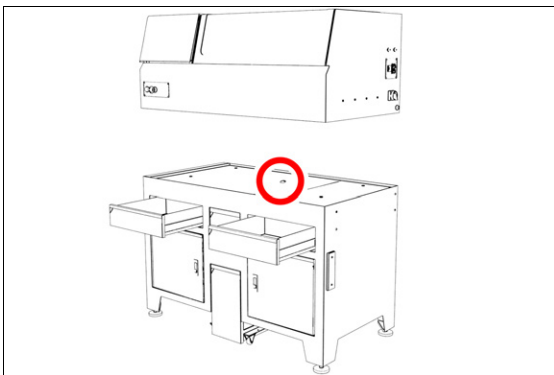


Figure 5-6: Route the coolant tube through the hole in the stand.

- Remove the four M10 lifting eyelets from the sides of the machine.
- Completely open each drawer on the stand. Then, identify the plastic tabs on the drawer slides. Push the tabs up or down to remove the drawer from the drawer slide.



Note: One tab gets pushed up, and the other pushed down.

- Install the nuts onto the bolts that secure the machine to the stand.
- Use an adjustable wrench to slowly lower the machine by adjusting the feet into the legs. Work your way around the machine, adjusting each foot a little at a time, being careful not to put the machine out of balance. Continue until the threaded rod on the feet is mostly inside the leg, the machine is stable, and the coolant drawer still fits under the stand.

Secure the Machine to a Workbench

- Remove the risers from the base of the machine.
- Install the machine feet in place of the risers that you removed in Step 1.
- If you're adding flood coolant to your system, attach the coolant tube onto the fitting on the bottom of the machine.

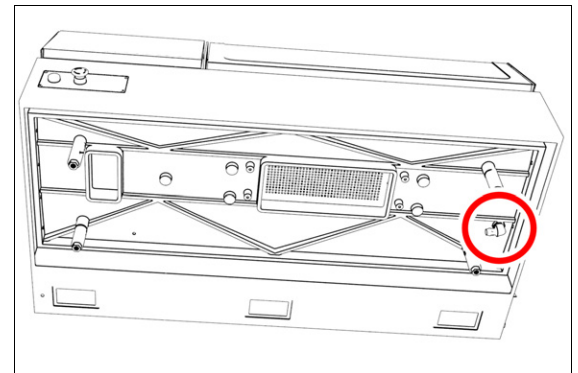


Figure 5-7: Coolant tube fitting on the bottom of the machine.

- Carefully lower the machine onto the 500-lb capacity workbench.
- Remove the four M10 lifting eyelets from the sides of the machine.
- Check the machine to make sure it doesn't rock back and forth.
 - If it moves, you must level the machine on the workbench. Go to Step 7.
 - If it doesn't move, you've completed the steps to secure the machine to the workbench.

5: INSTALLATION

5.6 Install Core Components

7. Identify the access panels on each end of the machine enclosure. Remove the button head cap screws with a 3 mm hex key. Then, set aside the screws and the access cover.
8. Loosen the jam butts on the leveling feet with a 19 mm wrench.
9. Use a wrench to thread the machine feet in or out to level the machine on the work bench.
10. Tighten the jam nuts on the leveling feet with a 19 mm wrench.
11. Use a 3 mm hex key to re-install the screws and access cover that you set aside earlier.

Note: When using the lathe with a desktop coolant system, it's important to periodically inspect the inside of the drip tray for accumulation of chips or other debris. This helps to prevent clogs of the drain line, which can result in coolant backing up into the machine's enclosure. For information on cleaning out the inside of the drip tray, go to "Clean the Coolant System" (page 172).



5.6.2 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.
3. Find the mounting pad on the machine stand.



Figure 5-8: Controller Arm mounting pad.

4. 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.

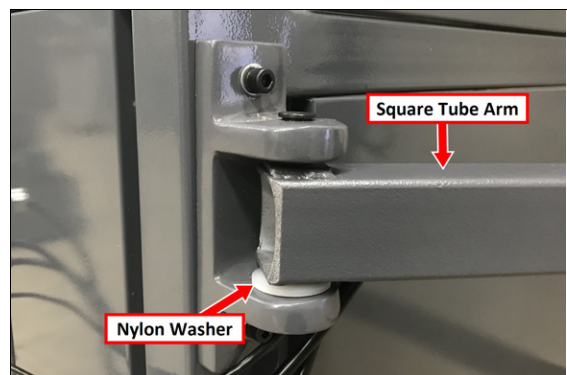


Figure 5-9: White nylon washer on the square tube arm.

5. 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.
6. 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.

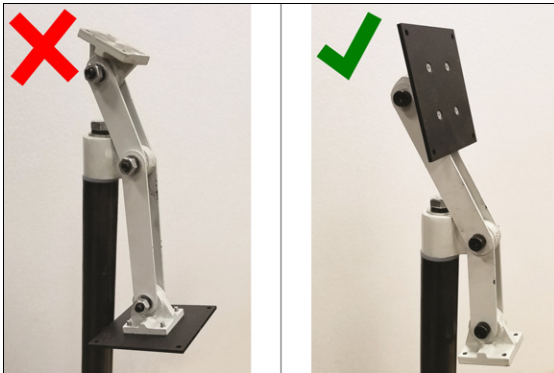


Figure 5-10: Incorrect and correct orientations of the monitor bracket on the Controller Arm.



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 5-11: 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.

5.6.3 Install the Controller

Depending on your machine configuration, do one of the following:

- If You Have a PathPilot Controller** Go to "Install the Monitor" (page 46).
- If you have a PathPilot Operator Console** Go to Set Up the PathPilot Operator Console.

Install the PathPilot Operator Console

Tools and Items Required

- 16 mm wrench
- Metric hex wrench set
- Phillips screwdriver

- Put the operator console against the monitor mounting plate and align the holes. Attach the operator console and monitor mounting plate together with four M4 × 12 mm socket head cap screws (provided with the operator console).

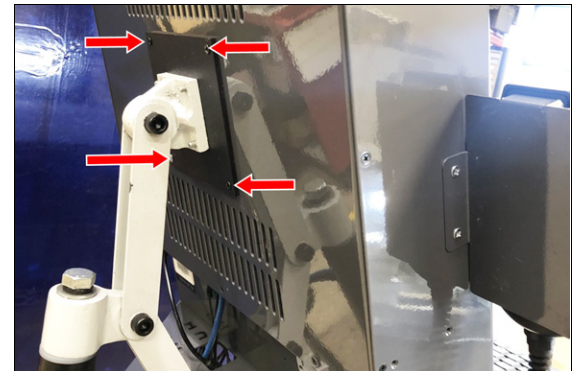


Figure 5-12: Operator console attached to the monitor mounting plate.

5: INSTALLATION

5.6 Install Core Components

2. Attach the keyboard tray to the lower controller arm mount with an 8 mm hex wrench.

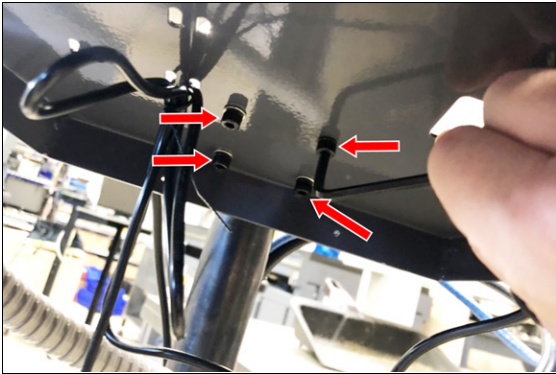


Figure 5-13: Attaching the keyboard tray to the lower controller arm mount (from below).

3. Adjust the operator console and the keyboard tray so that the two holes on the underside of the operator console line up with the holes on the keyboard tray.
4. Attach the keyboard tray to the operator console with two M3 Phillips screws (provided in the controller box) and a Phillips screwdriver.

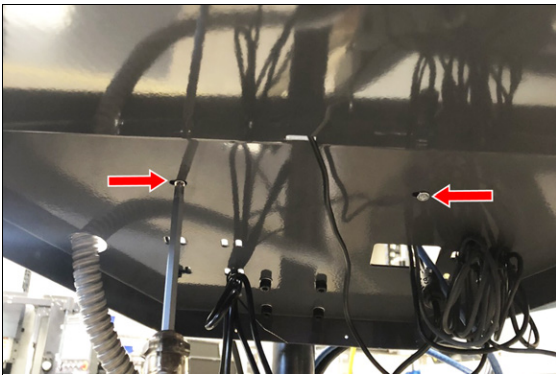


Figure 5-14: Attaching the keyboard tray to the operator console (from below).

5. Put the operator console's power supply into the power supply bracket (provided in the operator console box), and attach the assembly to the underside of the keyboard tray with M3 Phillips screws and a Phillips screwdriver.

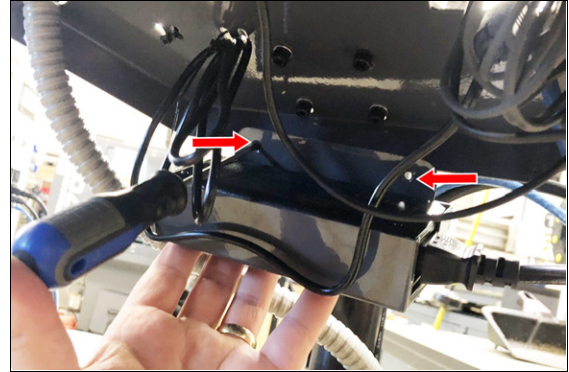


Figure 5-15: Attaching the operator console's power supply and power supply bracket to the keyboard tray (from below).

6. Connect the 12 ft power cable to the power supply.
7. Use the three pivot bolts on the controller arm monitor bracket to adjust the position of the operator console and the keyboard tray to your desired angle with a 3 mm hex wrench and 16 mm hex wrench.
8. If you have any of the following optional USB accessories, connect them to any **USB** port on the operator console:
 - Keyboard
 - Mouse
9. Connect the WiFi dongle to any **USB** port on the operator console.
10. Connect the Ethernet cable to the **Ethernet** port on the operator console.
11. Connect the barrel end of the power supply cable to the **Power Supply** port on the operator console.

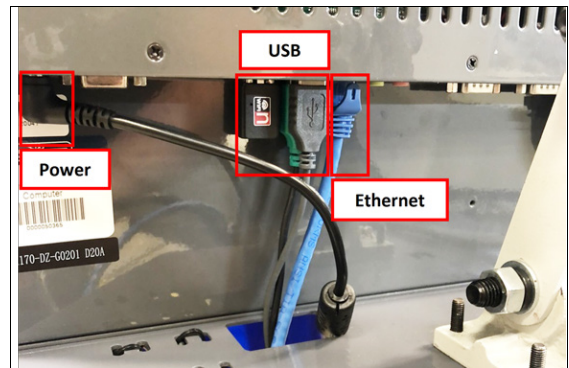


Figure 5-16: USB accessories connected.

12. Route the loose ends of the USB, Ethernet, and power supply cables through the square hole in the keyboard tray. Then, use the cable tie holes to secure the loose power supply and USB cables.

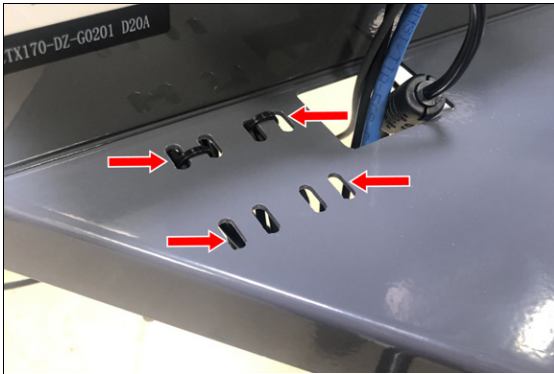


Figure 5-17: Cable tie holes.



Tip! If you're using a mouse, we recommend leaving some slack for it to move freely.

13. Connect the jog pendant cable and the Emergency Stop cable to the operator console. Route the loose end of the Emergency Stop cable through the round hole in the keyboard tray.

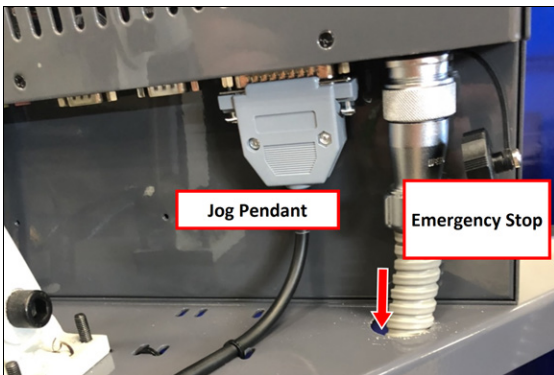


Figure 5-18: Jog pendant (left) and Emergency Stop (right) cables connected.

14. Route the loose end of the operator console's 12 ft power cable, the Emergency Stop cable, and the Ethernet cable down the controller arm. Then, route the cables through the slots in the square tube arm that's connected to the machine stand.
15. On the right side of the machine, connect the loose ends of the cables as follows:
 - a. Connect the operator console's power cable to any of the **Accessory** power outlets.

- b. Connect the Ethernet cable to the **Controller Communications** port.
 - c. Connect the Emergency Stop cable to the **Emergency Stop** port.

16. Secure the cables to the wire tie mounts that you installed on the round monitor post with four 4 in. cable ties.



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.

17. Open the electrical cabinet door.

5: INSTALLATION

5.6 Install Core Components

18. Depending on your machine's serial number, do one of the following:

- **LA10505 and Above:**

- a. Identify the 24-position terminal block in the lower center of the electrical cabinet. Remove the wire trough cover below it.
- b. The 24 terminal block positions are numbered from left to right, with position 1 beginning on the left. Identify position 18, and then identify the orange jumper wire labeled **440** (also connected to position 21).

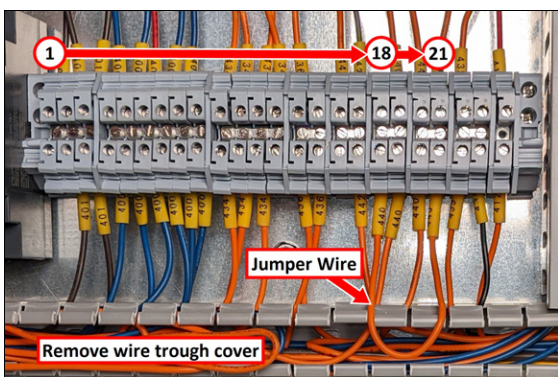


Figure 5-19: Identifying the jumper wire in the electrical cabinet.

- c. Remove the jumper wire from the machine.

- **LA10504 and Below** In the electrical cabinet, identify the jumper wire labeled **Remove to Install Console**, and remove it from the machine.

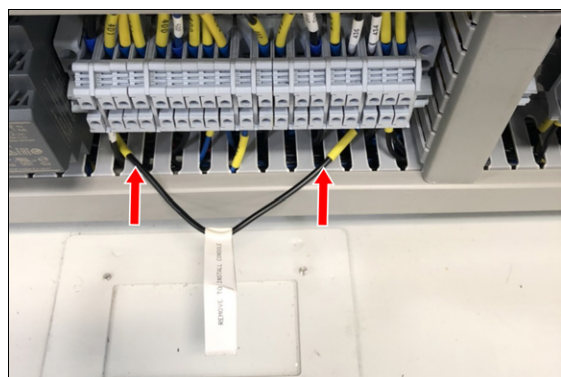


Figure 5-20: Jumper wire labeled Remove to Install Console.

⚠ WARNING! You must move the jumper from the machine. If the jumper isn't removed, the console's Emergency Stop button won't function.

19. Verify that the terminals are securely screwed down after removing the jumper wire.

20. Close the electrical cabinet door.

Install the Monitor

Tools and Items Required

- 3 mm hex wrench
- 8 mm hex wrench
- 16 mm wrench
- PathPilot Controller VESA Mount (PN 50382)

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).

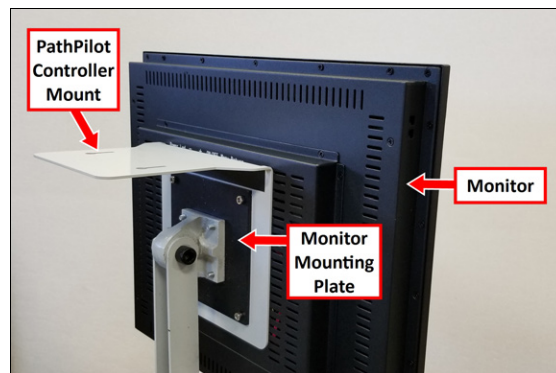


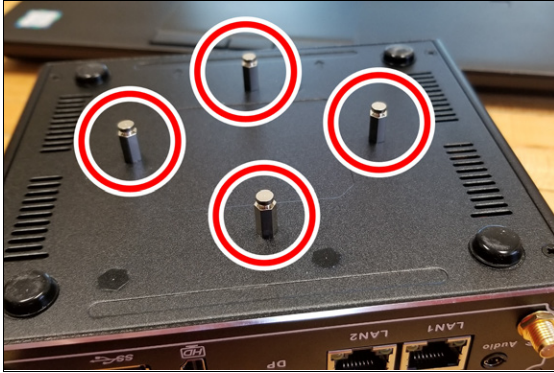
Figure 5-21: PathPilot controller mount attached to the back of the monitor.

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.

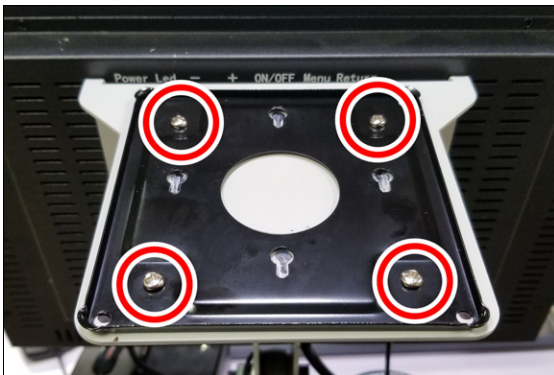
Install the PathPilot Controller

The PathPilot controller attaches to the top of the PathPilot Controller VESA Mount and behind the monitor.

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



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.



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



4. Connect all USB accessories to the controller:
 - [Jog Shuttle \(PN 30616\)](#) (Optional)
 - Keyboard
 - Mouse
 - Monitor
5. Connect the monitor's video cable to the PathPilot controller.
6. Connect the loose end of the video cable to the monitor.

5: INSTALLATION

5.6 Install Core Components

7. Connect the monitor's power cord. Depending on the type of monitor you have, do one of the following:
 - **Standard LCD Monitor (PN 30615)**
 - a. Connect two of the **Controller/Monitor** cables (provided in the machine owner's kit) together. Then, connect one end of the cable assembly into the monitor.
 - b. Route the loose end of the cable assembly down the monitor post, through the square tube arm, and toward the electrical cabinet.
 - c. Connect the cable assembly into one of the Accessory Power Ports.
 - **Touch Screen Kit (PN 35575)**
 - a. Connect the monitor's power supply into the monitor.
 - b. Route the power supply down the monitor post.
 - c. Connect one end of one **Controller/Monitor** cable (provided in the machine owner's kit) to the loose end of the monitor's power supply.
 - d. Route the loose end of the **Controller/Monitor** cable through the square tube arm and toward the electrical cabinet.
 - e. Connect the loose end of the **Controller/Monitor** cable into one of the Accessory Power Ports.
 - f. Attach the power supply to the monitor post with one cable tie (provided in the machine owner's kit).
8. Connect the PathPilot controller's power supply to the PathPilot controller.
9. Connect the loose end of the power supply to one of the **Controller/Monitor** cables provided.
10. Route the power cable down the monitor post, through the square tube arm, and toward the electrical cabinet.
11. Connect the loose end of the **Controller/Monitor** cable into one of the Accessory Power Ports.
12. Attach the PathPilot controller and monitor power supplies to the square tube arm with cable ties (provided in the machine owner's kit).

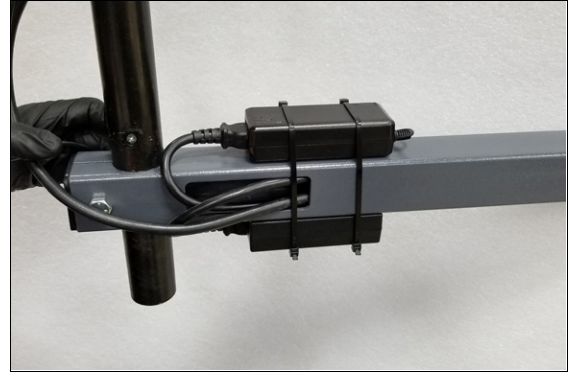


Figure 5-22: Power supply attached to the square tube arm.

13. Find the Ethernet cable (provided in the tool box), and then connect it to the Controller Communications outlet.
14. Route the loose end of the Ethernet cable toward the PathPilot controller, and then connect it to the PathPilot controller.
15. Secure the operator box cable, Ethernet cable, and power supply cables to the Controller Arm with four wire tie mounts and four cable ties.

5.7 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:

5.7.1 Power on the Machine	49
5.7.2 Verify Limit Switch Function	50
5.7.3 Verify Axes Function	50
5.7.4 Verify Spindle Function	51
5.7.5 Power off the Machine	52

5.7.1 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 5-23: The Main Disconnect switch on the side of the machine's electrical cabinet.

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

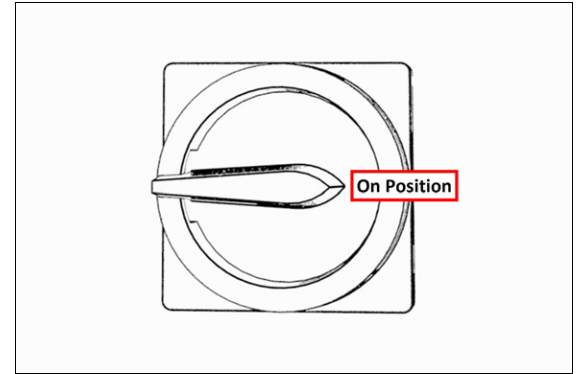


Figure 5-24: Example of the Main Disconnect switch in the On position.

Mains power is now connected to the machine.

5. Push the Power button on the PathPilot controller, if it's not already powered on.
6. On the **Tormach Machine Configuration** screen, verify that the version displayed in the lower right-hand corner is at least v2.7.0. If the version is older (or if no version is displayed), select **Update**.
7. Follow the on-screen instructions to configure the PathPilot operating system and PathPilot controller. When configuration is complete, the PathPilot operating system launches.



Note: After you first configure PathPilot, the operating system automatically launches whenever it's powered on.

8. Depending on which monitor you have, do one of the following:
 - a. **Standard Monitor** Go to the next step.
 - b. **Touch Screen Monitor** You must first make sure that the monitor is configured and calibrated. From the PathPilot interface, in the **MDI Line DRO** field, type `ADMIN TOUCHSCREEN`. Then, select the **Enter** key, and follow the on-screen instructions.

5: INSTALLATION

5.7 Verify the Installation

9. Rotate the Emergency Stop button one-quarter turn clockwise to release it.



Figure 5-25: Emergency Stop button.

10. Push the blue Reset button to enable the machine.



Figure 5-26: Reset button.

The axis drives are now powered on.

11. Verify that the blue Reset LED comes on. From the PathPilot interface, on the **Status** tab, verify that the **Machine OK** light changes from yellow to green. Once both are on, the machine is powered on and ready to operate.

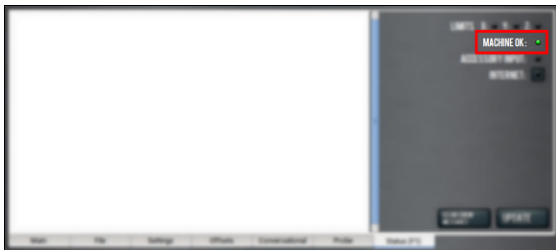


Figure 5-27: Machine OK light on the Status tab.

12. Select **Reset**.



Figure 5-28: Reset button.

This initializes the connection between the machine and controller.

5.7.2 Verify Limit Switch Function

You must confirm that the limit switches correctly operate.

1. From the PathPilot interface, on the **Status** tab, identify the **Limits** light.



Figure 5-29: Limits light on the Status tab.

2. Hold a piece of steel (like a screwdriver or a wrench) up to each limit switch.

The red light on each limit switch is on until the steel is within sensing range, at which point the red light dims or turns off.

From the PathPilot interface, on the **Status** tab, the **Limits** light comes on.

3. From the PathPilot interface, select **Reset**.

5.7.3 Verify Axes Function

You must confirm that the axes correctly operate.

1. Reference the axes: from the PathPilot interface, select **Ref Z**, **Ref X**.



Figure 5-30: Ref Z, Ref X, and Ref Y buttons.

The machine moves to the reference position.

2. Use the keyboard to verify axes motion:
 - Select the **Up Arrow** key and then the **Down Arrow** key.
The carriage moves away from you (X+), then toward you (X-).
 - Select the **Right Arrow** key and then the **Left Arrow** key.
The carriage moves right (Z+), then left (Z-).
3. If applicable, verify the optional Jog Shuttle:
 - Press any axis button on the Jog Shuttle — **X**, **Y**, **Z**, or **A** — to select an axis.

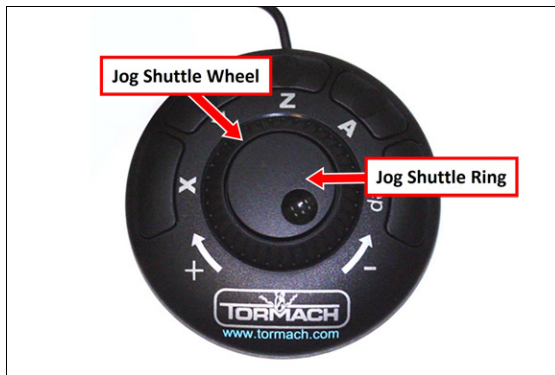


Figure 5-31: Functions on the optional Jog Shuttle.

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

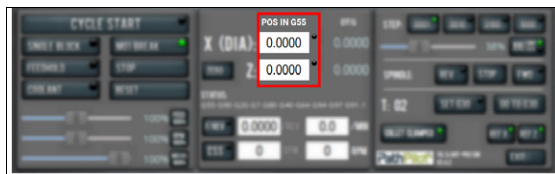


Figure 5-32: Axis lights.

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

Note: On this machine, two of the axis selection buttons — Y-axis and A-axis — won't result in any motion if selected.

5.7.4 Verify Spindle Function

You must confirm that the spindle correctly operates.

1. 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 5-33: 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 111).

2. From the PathPilot interface, in the **RPM DRO** field, type 180. Then select the **Enter** key.

Note: If you have an operator console, confirm that the **RPM** knob isn't set to 0%. If it is, turn the **RPM** knob.

3. Select **FWD**.

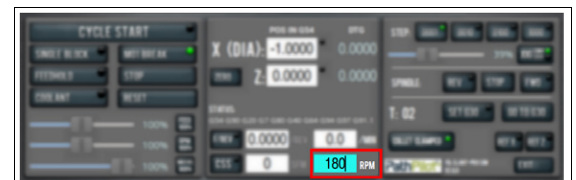


Figure 5-34: Spindle controls.

The spindle rotates clockwise at 180 rpm.

4. Examine the spindle direction: verify that it's rotating counterclockwise, as shown in the following image.



Figure 5-35: Spindle rotating counterclockwise, in the correct forward direction.

5: INSTALLATION

5.7 Verify the Installation

5. Select **STOP**.
The spindle stops rotating.
6. Select **REV**.
The spindle rotates clockwise at 180 rpm.
7. Examine the spindle direction: verify that it's rotating clockwise, as shown in the following image.

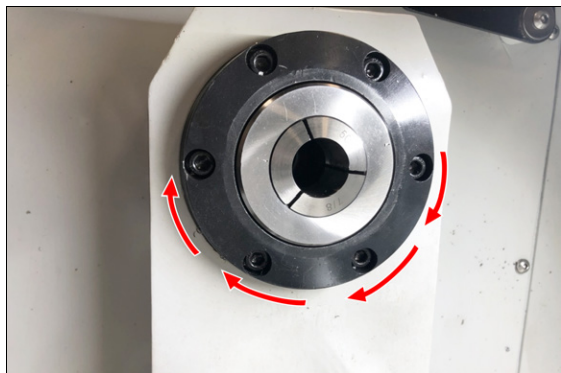


Figure 5-36: Spindle rotating clockwise, in the correct reverse direction.

8. Select **Stop**.
The spindle stops rotating.

5.7.5 Power off the Machine

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

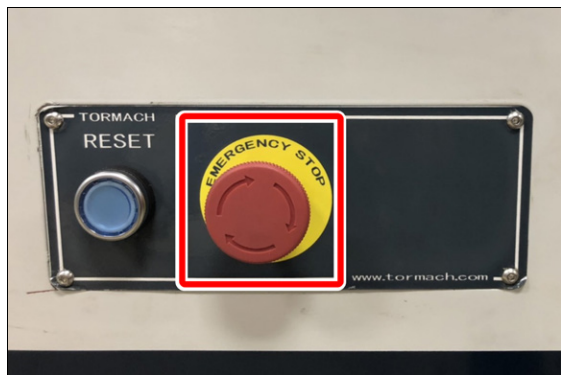


Figure 5-37: Emergency Stop button.

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

2. From the PathPilot interface, select **Exit**.
3. When prompted, select **OK**.
4. Once the PathPilot interface indicates that it's safe to

power off the machine, turn the Main Disconnect switch to **OFF**.

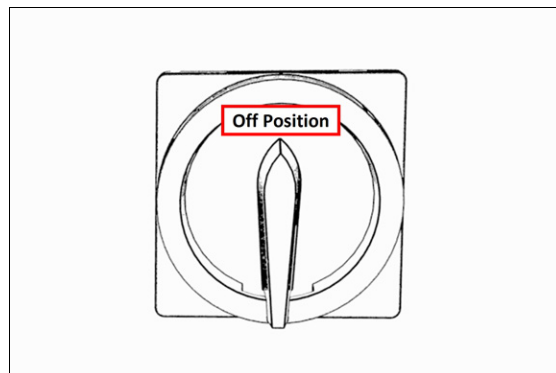


Figure 5-38: Example of the Main Disconnect switch in the Off position.

Mains power is disconnected from the machine.

5.8 INSTALL ACCESSORY COMPONENTS

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

5.8.1 Install the Machine Stand Coolant Kit (PN 50931) . . . 53

5.8.2 Set Up the Quick Change Tool Post Kit 53

5.8.1 Install the Machine Stand Coolant Kit (PN 50931)

1. Because the coolant pump is dual voltage, you must verify that the motor strapping is correct before you install it:
 - a. Remove the access cover on the coolant pump, and then identify the jumper configuration diagram.

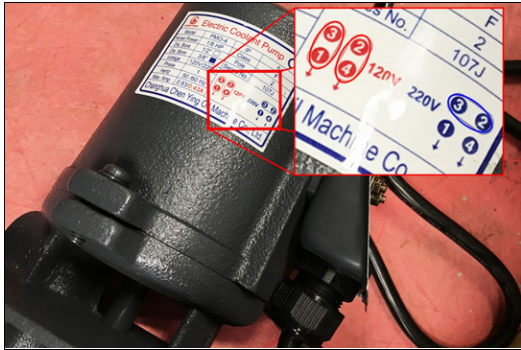


Figure 5-39: Coolant pump motor jumper configuration diagram.

- b. Verify that the jumpers are in the correct configuration (120 Vac or low volt) for your installation. If necessary, adjust the configuration. Go to "Adjust the Coolant Pump Motor Strapping" (below).
2. Remove the four M5 x 0.8 - 16 socket head cap screws pre-installed on the coolant tank. Set the screws aside.
3. Put the coolant pump into the coolant tank, and secure it with the four socket head cap screws that you set aside in Step 2.
4. Attach the provided 90° push-to-connect fitting to the coolant pump.
5. Find the coolant tube, which you routed through the machine stand earlier. Attach the loose end of the coolant tube to the push-to-connect fitting on the coolant pump.
6. Route the power cord on the coolant pump to the right side of the machine, and connect it to the Coolant Pump Power outlet.

7. Verify that the coolant setup operates properly:
 - a. From the PathPilot interface, on the **Main** tab, select **Coolant**.
The coolant pump powers on.
 - b. Select **Coolant** again.
The coolant pump powers off.

Adjust the Coolant Pump Motor Strapping

Because the coolant pump is dual voltage, you may need to adjust the motor strapping.

1. Identify the jumper configuration diagram.



Figure 5-40: Coolant pump motor jumper configuration diagram.

2. Remove the access cover on the coolant pump.
3. Reference the configuration diagram and move the jumper to the 120 Vac (low volt) position. Don't change the position of any wires – just the jumper that connects the terminals.

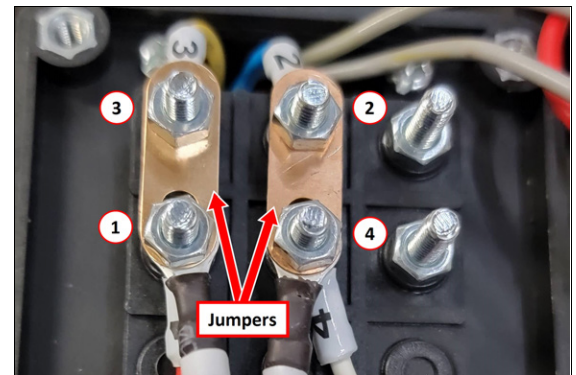


Figure 5-41: Coolant pump motor strapping adjusted for 120 Vac.

4. Reinstall the access cover on the coolant pump.

5.8.2 Set Up the Quick Change Tool Post Kit

Complete the following steps in the order listed:

Required Tools 54

5: INSTALLATION

5.8 Install Accessory Components

Install the Tool into the Quick-Change Tool Holder.....	54
To Use a Boring Bar.....	54
Align the Tool.....	55

Required Tools

- Adjustable wrench
- Dead-blow hammer
- Magnetic dial test indicator
- Metric hex wrench set
- Rust preventative

Install the Tool into the Quick-Change Tool Holder

1. Find a tool and its equivalent tool holder, and put the tool into the tool holder. Lock the tool into the tool holder: with the tool pointing toward you, tighten the hex head screws.

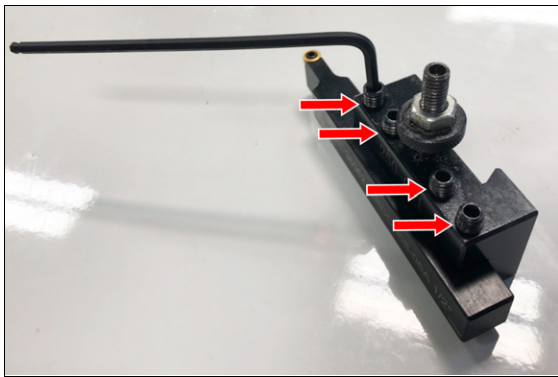


Figure 5-42: A tool with its equivalent tool holder.

2. Set the rough tool height by adjusting the thumb screw. Then, secure it in place with the stop nut. You'll make fine adjustments later in this procedure to bring the tool plane on center with the spindle.

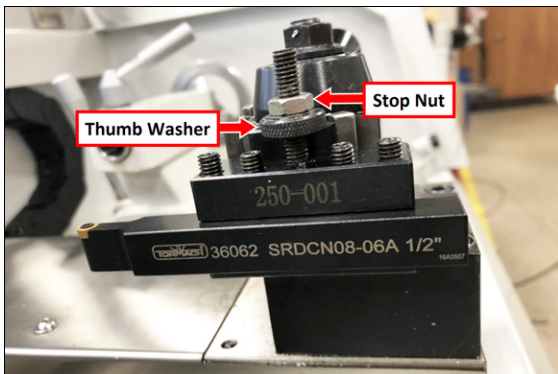


Figure 5-43: Adjustment components on the tool holder.

3. To attach the tool holder, the tool post must be in the unlocked position: move the tool post handle up.

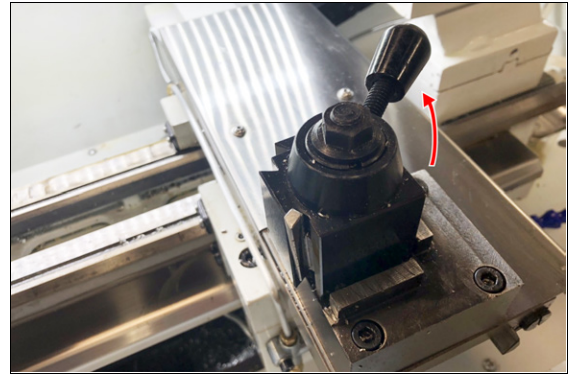


Figure 5-44: Quick change tool post positions.

4. Put the tool holder onto the tool post, and then move the tool post handle down to the locked position.
5. Before operating the machine, you must make sure that the quick change tool holder is aligned. Go to [Align](#).

To Use a Boring Bar

1. To attach the tool holder, the tool post must be in the unlocked position: move the tool post handle up.
2. Put the tool into the boring bar holder.
3. Put the tool holder onto the tool post, and then move the tool post handle down to the locked position.
4. Lock the tool into the tool holder: tighten its two hex head screws.

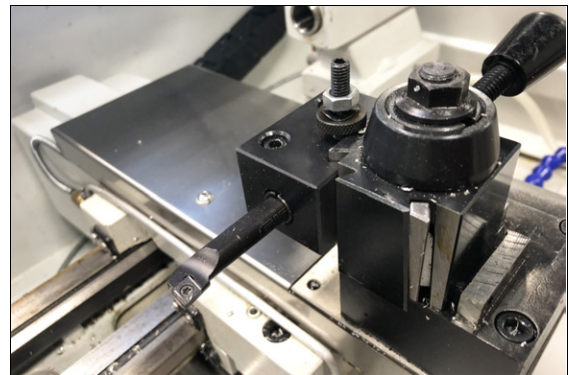


Figure 5-45: Boring bar installed in a boring bar tool holder.

Tip! When you remove the tool, loosen the hex head screws, and gently tap them with a dead-blow hammer.

Align the Tool

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 machine's Reset button (next to the Emergency Stop button).
 - d. Bring the machine out of reset and reference it.
2. Attach a magnetic dial test indicator to the spindle.
3. Put the indicator tip on the face of the tool holder.

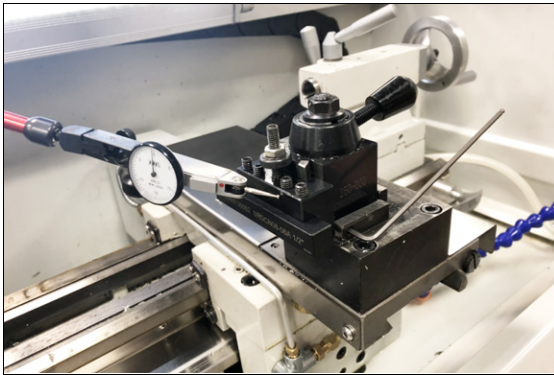


Figure 5-46: Magnetic dial test indicator on the face of the tool holder.

4. Slowly jog the machine along the X-axis to move the indicator tip along the face of the tool.
5. Adjust the quick change tool post alignment as necessary: use a 3 mm hex wrench to adjust the two set screws on the front and right sides of the quick change tool post risers to clock the tool post until the indicator reads zero.

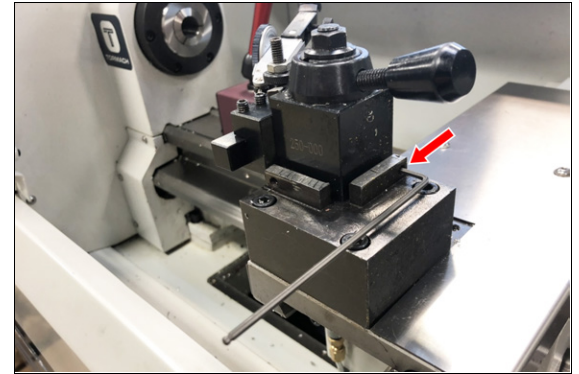
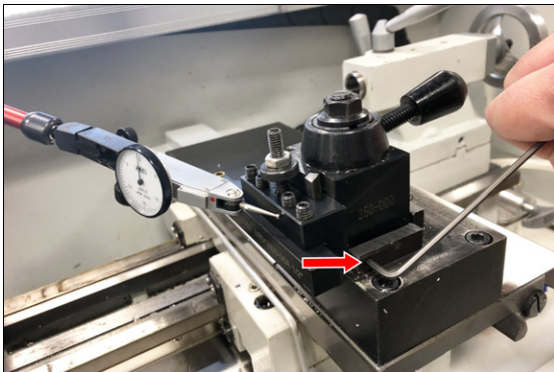


Figure 5-47: Aligning the quick change tool post.

6. Securely tighten the tool post nut with an adjustable wrench.

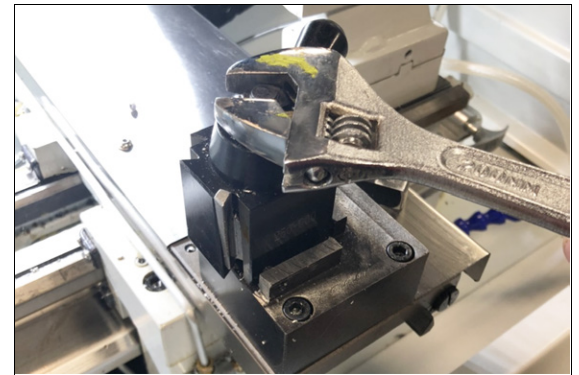


Figure 5-48: Tool post nut securely tightened with an adjustable wrench.

7. Repeat Steps 4 through 5 to make sure that the tool post is still correctly aligned after tightening the tool post nut.

5: INSTALLATION

5.9 Set Up the PathPilot Controller

5.9 SET UP THE PATHPILOT CONTROLLER

Before operating your machine, configure in PathPilot the date, time, keyboard language, and — if applicable — the optional Touch Screen Kit (PN 35575).

5.9.1 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**.

5.9.2 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**.

5.9.3 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.

5.9.4 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**.

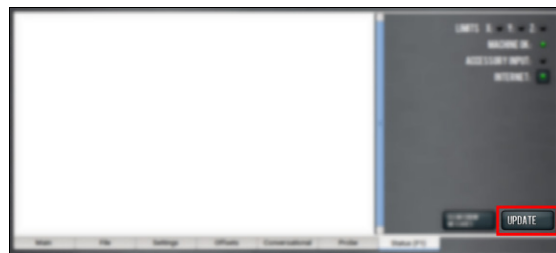


Figure 5-49: Update button on the Status tab.

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

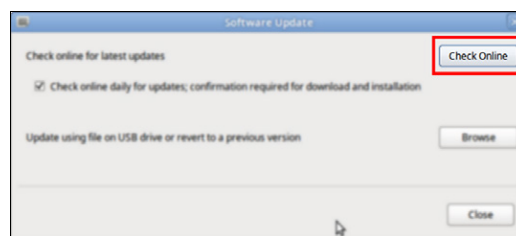


Figure 5-50: Software Update dialog box.

4. Select **Install**.

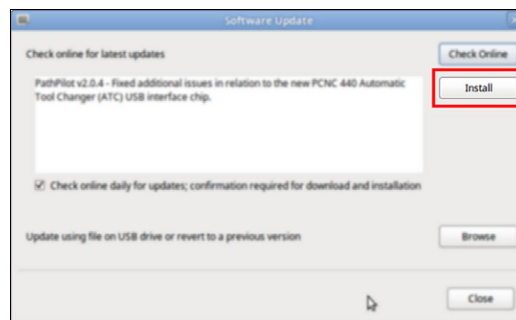


Figure 5-51: Install button on the Software Update dialog box.


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

6.1 System Reference.....	58
6.2 Basic Controls Reference.....	59

6: SYSTEM BASICS

6.1 System Reference

6.1 SYSTEM REFERENCE

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

6.1.1 Spindle

The machine spindle holds a workpiece using one of the following methods:

1. A 5C taper insert used with 5C collets for precision turning of stock less than 1.125 in. in diameter.
2. The spindle nose D1-4 cam lock system with D1-4 cam lock chucks for precision turning of larger stock.

- **Spindle Power** 1.5 hp (1.11 kW)
- **Spindle Speed** Two spindle ranges with speeds from 180 to 5,000 rpm

6.1.2 Axes

The machine has two linear axes of motion used for machining:

- The Z-axis, which is parallel to rotational axis of the spindle.
The -Z direction is toward the headstock, and the +Z direction is opposite the headstock (toward the tailstock).
- The X-axis, which is perpendicular to the Z-axis and nominally parallel to the long dimension of the carriage.
The +X direction is toward the front, and -X direction is toward the rear.
- **X-Axis** 4.5" (114 mm)
- **Z-Axis** 10" (254 mm) with tailstock

6.2 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.

6.2.1 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 Reset button and the Emergency Stop button, located on the front left side of the base of the machine. 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 Reset button to enable the machine, allowing spindle and axis motion. The Reset button's LED turns on when the machine is enabled and the spindle and axis drives receive power.

6.2.2 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 and Monitor** Shows the PathPilot interface (monitor, touch screen, or integrated console display)
- **Input Devices** Keyboard and mouse for navigation and data entry

- **Manual Control** Jog pendant or jog shuttle for manual axis movement

The specific hardware configuration depends on your machine model.



PATHPILOT INTERFACE OVERVIEW

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

7.1 About PathPilot	62
7.2 Notebook Section	63
7.3 Persistent Controls	65
7.4 Keyboard Shortcuts	67
7.5 Manage PathPilot Versions	68

7: PATHPILOT INTERFACE OVERVIEW

7.1 About PathPilot

7.1 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.

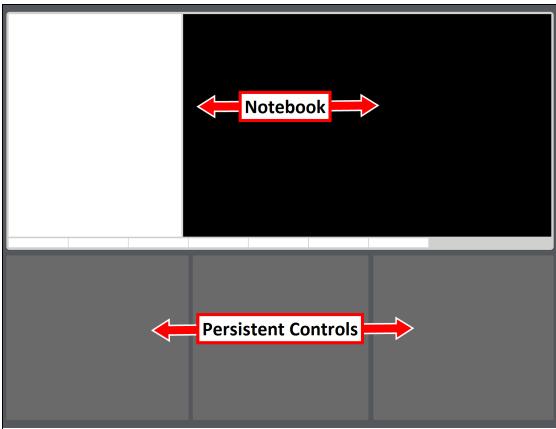


Figure 7-1: Sections in the PathPilot interface.

7.2 NOTEBOOK SECTION

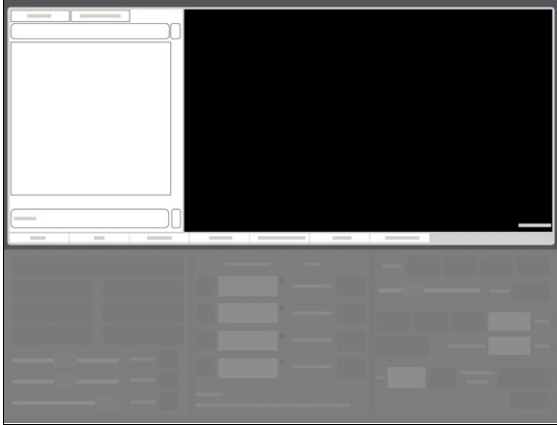


Figure 7-2: Notebook section.

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

7.2.1 Main Tab	63
7.2.2 File Tab	63
7.2.3 Settings Tab	63
7.2.4 Offsets Tab	64
7.2.5 Conversational Tab	64
7.2.6 Status Tab	64

7.2.1 Main Tab



Figure 7-3: 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 73); "Close the Current Program" (page 73).
- Send G-code commands directly to the machine using the Manual Data Input (MDI) Line DRO field.
For information, see "Manually Enter Commands" (page 115).
- 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 75); "Expand the G-Code Tab" (page 74); "Change the View of the Tool Path Display" (page 76).

- Make and restore backup files of your settings.
For information, see "Create Backup Files" (page 121); "Restore Backup Files" (page 122).

7.2.2 File Tab



Figure 7-4: 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 133).
- Edit G-code files.
For information, see "Edit G-Code" (page 73).
- Load .nc files into PathPilot to run a program.
For information, see "Load G-Code" (page 133).
- Move files within the system.
For information, see "Preview G-Code Files" (page 72); "Manage System Files" (page 121).

7.2.3 Settings Tab

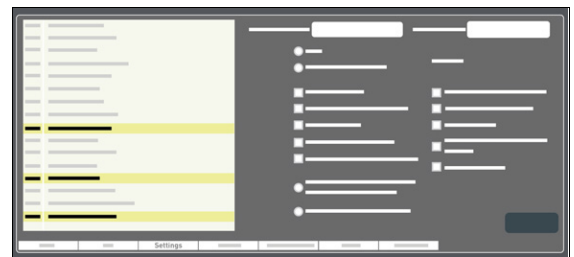


Figure 7-5: Settings tab.

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 94).

7: PATHPILOT INTERFACE OVERVIEW

7.2 Notebook Section

- Change the screen's layout orientation (landscape or portrait).
For information, see "Change the Screen Orientation" (page 94).
- Configure PathPilot for the accessories you're using.
For information, see "Enable the On-Screen Keyboard" (page 97); "Enable the USB M-Code I/O Interface Kit" (page 98); "Use a USB Camera" (page 98).
- Turn on feeds and speeds suggestions when using conversational programming.
For information, see "Enable Feeds and Speeds Suggestions in Conversational Routines" (page 117).
- Specify the way in which you want to use a G30 move.
For information, see "Limit G30 Moves" (page 97).
- Identify the available G-code modes that you can use.
For information, see "View Available G-Code Modes" (page 104).

7.2.4 Offsets Tab

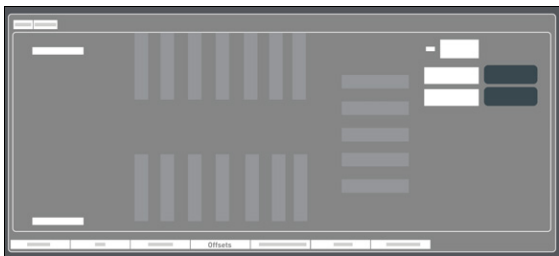


Figure 7-6: Offsets tab.

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

- Import and export .csv files of your tool table.
For information, see "Import and Export the Tool Table" (page 137).
- Work with a table of tool descriptions and tool offsets.
For information, see "Set Tool Geometry Offsets" (page 135).
- Read the currently programmed work offsets.
For information, see "View Work Offsets" (page 104).
- Change the tool number.
For information, see "Change the Tool Number" (page 114).

7.2.5 Conversational Tab

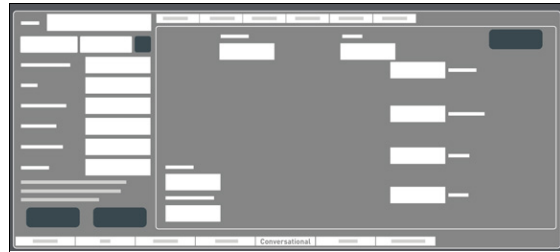


Figure 7-7: Conversational tab.

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 77).

7.2.6 Status Tab



Figure 7-8: 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 93).
- Update or install a previous version of PathPilot.
For information, see "Manage PathPilot Versions" (page 68).

7.3 PERSISTENT CONTROLS



Figure 7-9: 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:

7.3.1 Program Control Area	65
7.3.2 Position Status Area	65
7.3.3 Manual Control Area	65

7.3.1 Program Control Area



Figure 7-10: 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 127).
- Start, stop, or pause a G-code program.
For information, see "Start a Program" (page 107); "Stop Machine Motion" (page 107); "Use the Feed Hold Function" (page 109).
- Use overrides to change the feed rate, spindle speed, and maximum velocity.
For information, see "Use the Feed Rate Override Function" (page 109); "Use the Maxvel Override Function" (page 110); "Use the Spindle Override Function" (page 110).

- Manually control a G-code program.
For information, see "Use M01 Break Mode" (page 110); "Use Single Block Mode" (page 110).

7.3.2 Position Status Area



Figure 7-11: Position Status area.

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

- Create work offsets.
For information, see "Set Work Offsets" (page 134).
- Understand how you're jogging the machine.
For information, see "View the Active Axis to Jog" (page 105); "View the Current Machine Position" (page 106); "View the Distance to Go" (page 108).
- Quickly determine which G-code modes are active.
For information, see "View the Active G-Code Modes" (page 108).
- Change the spindle speed or feed rate.
For information, see "Change the Feed Rate Mode" (page 112); "Change the Spindle Speed Command Mode" (page 113).

7.3.3 Manual Control Area



Figure 7-12: 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 129).
- Preset or go to a G30 position.
For information, see "Use a G30 Position" (page 114).
- Reference the machine axes.
For information, see "Reference the Machine" (page 128).

7: PATHPILOT INTERFACE OVERVIEW

7.3 Persistent Controls

- Manually start and stop the spindle.
For information, see "About Spindle Controls"
(page 113).
- Clamp and unclamp an automatic collet closer.

7.4 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+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

7: PATHPILOT INTERFACE OVERVIEW

7.5 Manage PathPilot Versions

7.5 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)

7.5.1 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**.

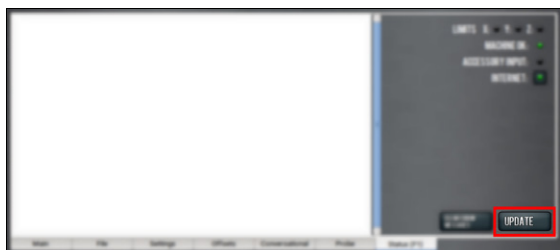


Figure 7-13: Update button on the Status tab.

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

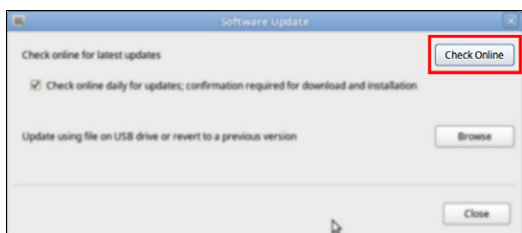


Figure 7-14: Software Update dialog box.

4. Select **Install**.

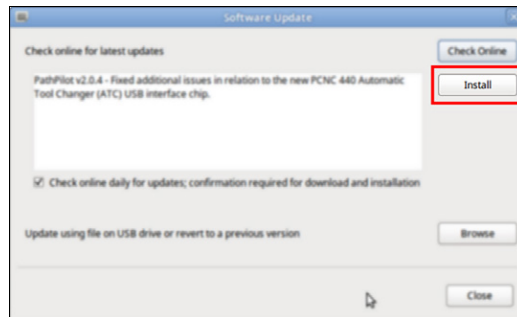


Figure 7-15: Install button on the Software Update dialog box.

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.

7.5.2 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**.

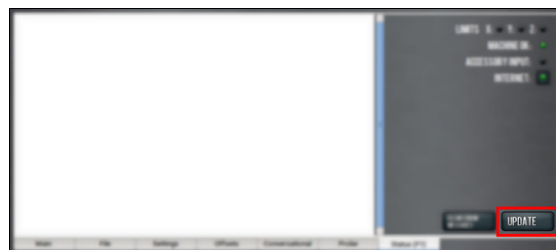


Figure 7-16: Update button on the Status tab.

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

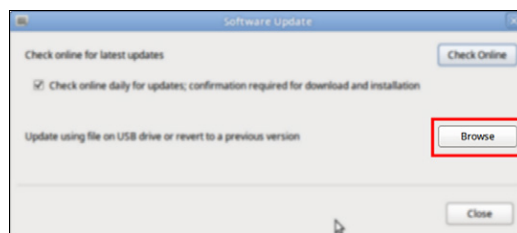


Figure 7-17: Software Update dialog box.

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



Figure 7-18: Browse dialog box.

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

7.5.3 Install a Previous Version of an Update File

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

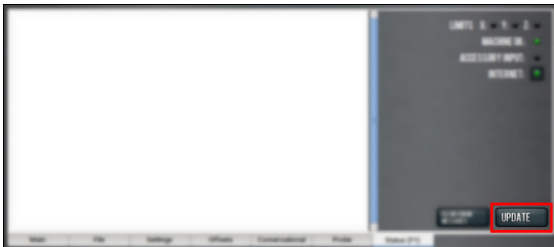


Figure 7-19: Update button on the Status tab.

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

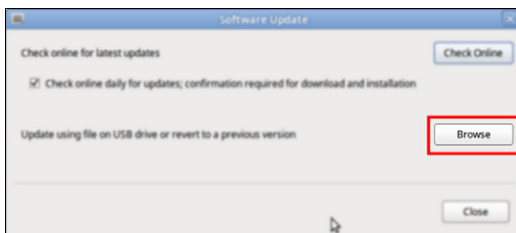


Figure 7-20: Software Update dialog box.

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

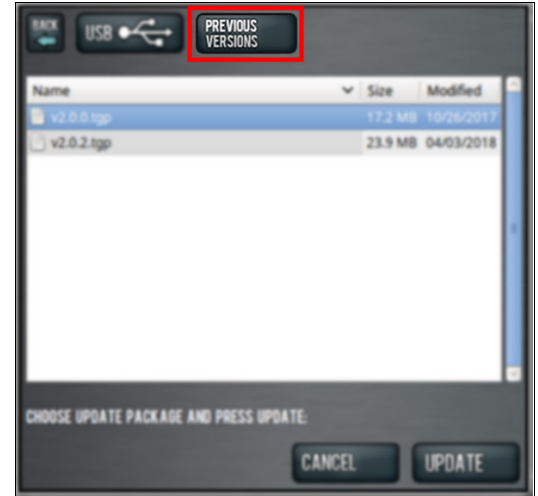


Figure 7-21: Browse dialog box.

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



PATHPILOT TOOLS AND FEATURES

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

8.1 Create and Load G-Code Files	72
8.2 Machine Settings and Accessories	93
8.3 Set Up G-Code Programs	102
8.4 Run G-Code Programs	105
8.5 Control G-Code Programs	109
8.6 System File Management	121

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

8.1 CREATE AND LOAD G-CODE FILES

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

8.1.1 Load G-Code	72
8.1.2 Edit G-Code	73
8.1.3 Read G-Code	74
8.1.4 Use Conversational Programming	77

8.1.1 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" (page 133).

To load G-code:

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

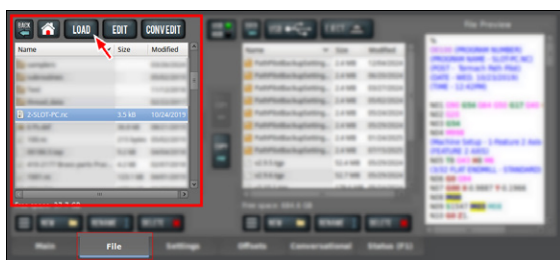


Figure 8-1: Controller Files window on the File tab.



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. You can either use a USB drive or PathPilot HUB (our cloud-based simulator) to transfer files. For more information on PathPilot HUB, go to hub.pathpilot.com.

To transfer files to and from the controller:

1. Either insert a USB drive into any open USB port, or sign in to PathPilot HUB.
2. From the **File** tab, select the file to transfer (either in the **USB / HUB Files** window or the **Controller Files** window).

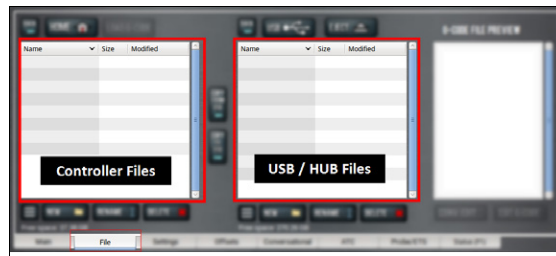


Figure 8-2: File tab.



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 ←** or **Copy →**.

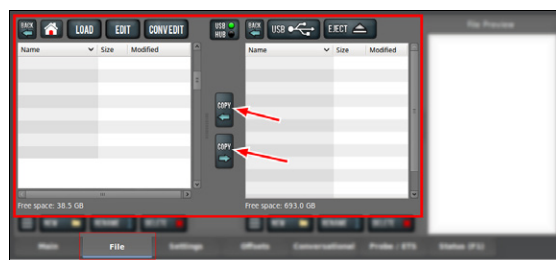


Figure 8-3: File tab.



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. If you're using a USB drive, select **Eject**.
It's safe to remove the USB drive from the controller.

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.

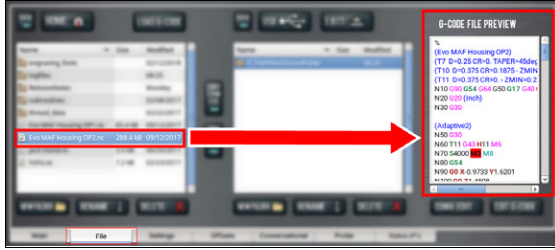


Figure 8-4: File tab.

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.

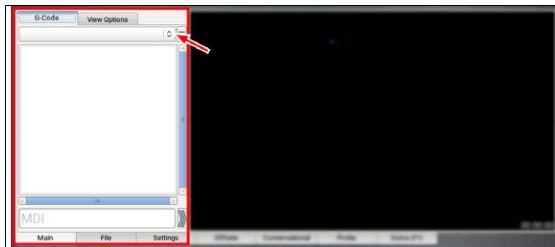


Figure 8-5: Recent Files menu on the Main tab.

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**.

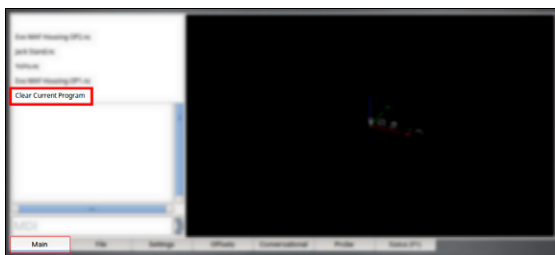


Figure 8-6: Recent Files menu on the Main tab.

The currently loaded G-code program closes.

8.1.2 Edit G-Code

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

- **Edit G-Code with a Text Editor** 73
- **Edit G-Code with Conversational Programming** 73

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" (page 133).

To edit G-code with a text editor:

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

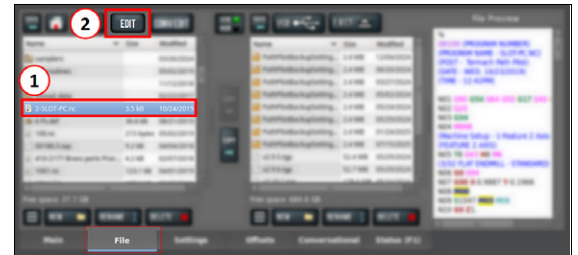


Figure 8-7: Edit G-code button on the File tab.

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" (page 133).

To edit G-code with conversational programming:

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

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.

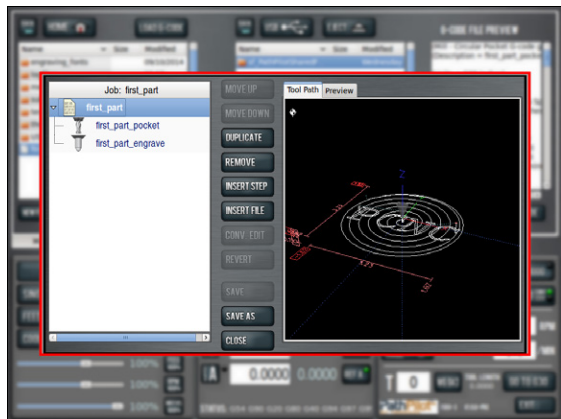


Figure 8-8: Job assignment editor.

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

Change the Step Order.....	74
Create a New Job Assignment.....	74
Load an Existing G-Code File.....	74
Edit a Job Assignment.....	74

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**.

8.1.3 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.....	74
Search in the Code.....	75
Set a New Start Line.....	75
Change the View of the Tool Path Display.....	76

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**.

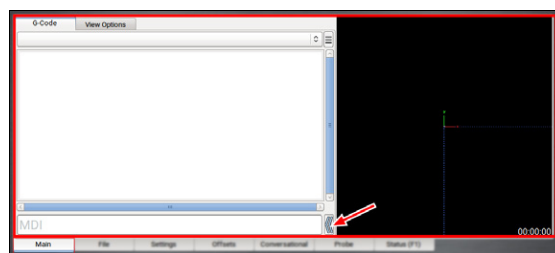


Figure 8-9: Window Expander on the Main tab.

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" (on the previous page).

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" (below).
- **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.

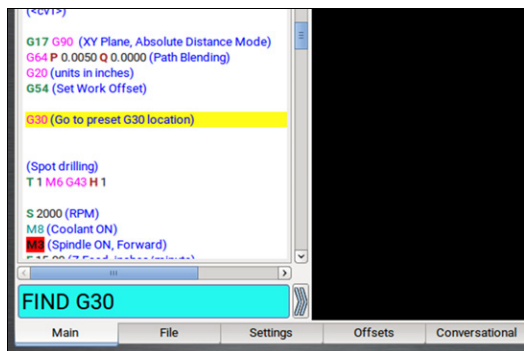


Figure 8-10: Search for a text command.

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

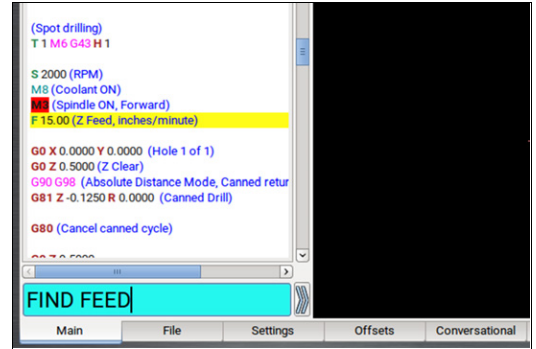


Figure 8-11: Search for a feed command.

- **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:

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

- Right-click any line in the program.

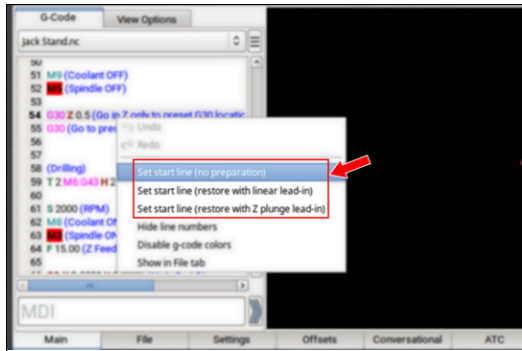


Figure 8-12: Accessing the Options menu by right-clicking.

- 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.

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 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 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 loaded). This option doesn't require you to jog to the exact lead-in position.

Change the View of the Tool Path Display

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

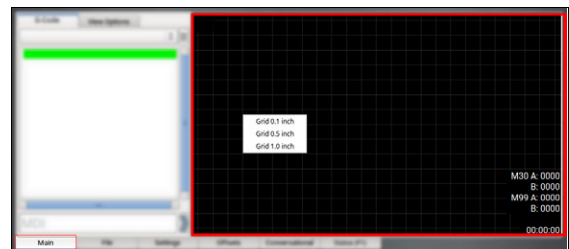


Figure 8-13: Tool Path display on the Main tab.

- Select the **View Options** tab.

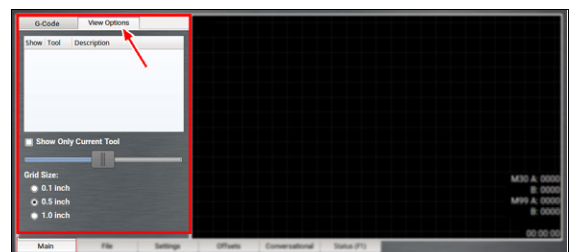


Figure 8-14: View Options tab on the Main tab.

2. Select a new view.
For information, see "About the Tool Path Display" (on the next page).

About the Tool Path Display

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

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** 10 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.

8.1.4 Use Conversational Programming

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

About Conversational Programming	77
Create an Outside Diameter	77
Create an Internal Diameter	79
Create a Profile on a Part	82
Create a Face on a Part	85
Create a Chamfer or Radius on a Part	86
Create a Groove or Part a Workpiece	87
Create Holes on a Part or Tap a Hole	89
Create Threads on a Part	91

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 an Outside Diameter

Using conversational programming, you can program PathPilot to rough and finish three features: an outside diameter, a fillet (corner radius), or an adjacent face. For information, see "About OD Turning" (page 79).

Before You Begin

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

- The value used in the **Z End** DRO field should be less than the value used in the **Z Start** DRO field.
- The value used in the **Fillet Radius** DRO field should be larger than the radius of the tool.
- The tool is cutting both an outside diameter and a face — valid tools are limited to orientation 2 for a front tool.
- The face is always on the headstock end of the diameter being cut.
- The fillet calculation doesn't use cutter radius compensation: the middle of the fillet isn't on the true radius for a tool with a tip radius.

To create an outside diameter:

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

1. From the **Conversational** tab, select the **OD Turn** tab.
2. From the **Conversational DROs** group, set the parameters for the OD turning operation.

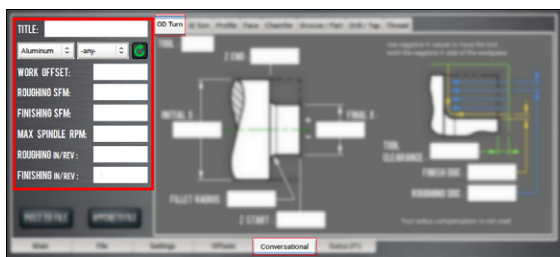


Figure 8-15: Conversational DROs on the OD Turn tab.

3. Work through the program-specific DRO fields:
 - a. In the **Tool** DRO field, type the currently selected tool as it's defined in the **Tool Table** window (on the **Offsets** tab).

This DRO field is a command value — it sets the tool number for a tool change at the start of the program.
 - b. In the **Initial X** DRO field, type the value of the stock's outside diameter.



Note: This DRO field is a reference value. It's also used with the **Tool Clearance** DRO field to locate some of the transitions between rapid and feed rate. If the values in the **Initial X** DRO field and the **Final X** DRO field are both positive, the tool works on the positive X side of the spindle center (the side toward you). If they're both negative, the tool works the negative side of the spindle (the side away from you). It's an error if there's a positive and a negative value for each DRO field.

- c. In the **Final X** DRO field, type the desired value of the part's final outside diameter.
- d. In the **Z Start** DRO field, type the location of the stock's face.



Note: This DRO field is used with the **Tool Clearance** DRO field to set the transition between rapid and feed rate on some Z moves.

- e. In the **Z End** DRO field, type the desired location of the part's face.
- f. In the **Fillet Radius** DRO field, type the desired radius between the part's outside diameter and its face. For no radius, type 0.



Note: If you type a value that's less than the tip radius, PathPilot drives the cutter to the corner. If you type a value that's larger than the Z range (the distance between the location of the stock's face and the desired location of the part's face) or the X range (half of the distance between the stock's outside diameter and the desired value of the part's outside diameter), the fillet starts or ends outside of the stock perimeter, and it doesn't end at the specified X and Z locations.

- g. In the **Tool Clearance** DRO field, type the distance required for clearance when the machine makes rapid movements between the stock's outside diameter its face. Because there's only one value used for X and Z moves, use the greater of the two clearances.



Note: Use larger values to begin; once you're familiar with how the program works, smaller values may save time. This DRO field is also sometimes used as a location for retracting the tool while making cutting passes.

- h. In the **Roughing DOC** DRO field, type the desired amount of material to remove from the radius of the stock on each roughing pass. The depth of cut is adjusted to get the value used in the G-code.

- i. In the **Finish DOC** DRO field, type the desired amount of material required for one finish pass (completed after roughing).

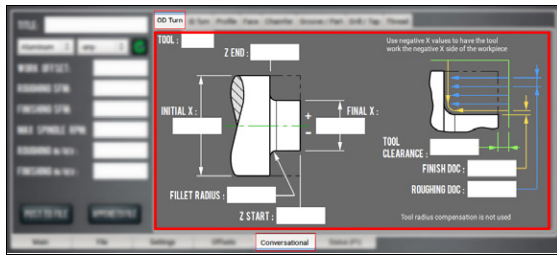


Figure 8-16: Program-specific DROs on the OD Turn Tab.

About OD Turning

Outside diameter turning is the process of removing material on the outside of a part.

OD Turning in PathPilot

During an OD turning routine, PathPilot does the following:

1. Roughing starts at the location typed in the **Initial X** DRO field, and incrementally cuts diameters at an adjusted depth of cut using the value typed in the **Roughing DOC** DRO field.
2. The finish diameter is started at the following location: (Final X + [2 × Finish DOC]). At this point, a single finishing pass is done at the value typed into the **Finish DOC** DRO field.

The finish pass starts at the +Z (tailstock) end of the outside diameter and feeds to the middle of the fillet.



Note: Since there is only one finish pass, the value in the **Finish DOC** DRO field isn't adjusted.

3. The tool retracts to the stock diameter.
4. The face finish pass is cut from the stock diameter to the end of the fillet.

Create an Internal Diameter

Using conversational programming, you can program PathPilot to cut a basic or extended internal diameter on a part. For information, see "About ID Turning" (page 81).

Before You Begin

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

- Valid tool orientations are limited to type 3 for a front tool.
- The tool path changes by 90° on the same side of the tool, so a form tool (narrow tip angle) and separate roughing DOCs are needed.

Basic Internal Diameters

To create a basic internal diameter:

1. From the **Conversational** tab, select the **ID Turn** tab.
2. From the **Conversational DROs** group, set the parameters for the ID turning operation.

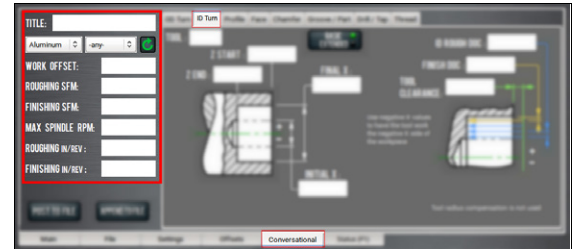


Figure 8-17: ID Turn tab on the Conversational tab.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

3. Work through the program-specific DRO fields:
 - a. In the **Tool** DRO field, type the currently selected tool as it's defined in the **Tool Table** window (on the **Offsets** tab).

This DRO field is a command value — it sets the tool number for a tool change at the start of the program.
 - b. In the **Initial X** DRO field, type the diameter of the pilot hole. Make sure that the diameter is large enough to clear the tool holder's X width.
 - c. In the **Final X** DRO field, type the desired final diameter of the internal diameter.
 - d. In the **Z Start** DRO field, type the location of the stock's face.



Note: This DRO field is used with the **Tool Clearance** DRO field to set the transition between rapid and feed rate on some Z moves.

- e. In the **Z End** DRO field, type the desired final location for the part's face.
- f. In the **Tool Clearance** DRO field, type the distance required to retract the tool and transition between rapid and cutting feed rate. Because there's only one value used for X and Z moves, use the greater of the two clearances.



Note: Use larger values to begin; once you're familiar with how the program works, smaller values may save time. Larger values bring the back of the tool holder closer to the ID wall on the end of facing cuts.

- g. In the **ID Rough** DRO field, type the depth of material to cut on the radius of the bore. The depth of cut is adjusted to get the value used in the G-code.
- h. In the **Finish DOC** DRO field, type the desired amount of material required for one finish pass on the ID, fillet, and face (completed after roughing).

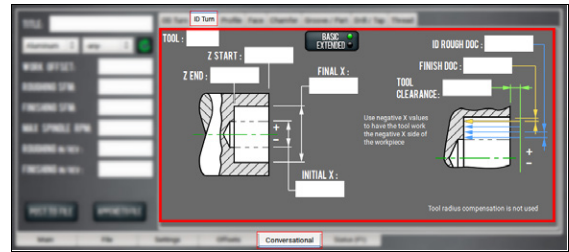


Figure 8-18: Program-specific DROs on the ID Turn tab.

Extended Internal Diameters

To create an extended internal diameter:

1. From the **Conversational** tab, select the **ID Turn** tab.
2. Select the button to toggle from Basic to Extended mode.

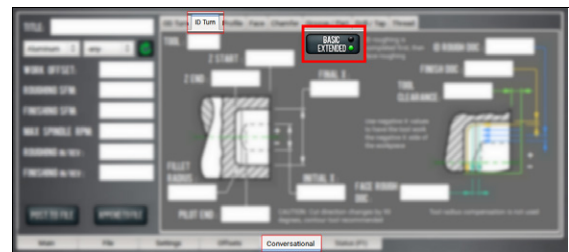


Figure 8-19: ID Turn tab in Extended mode.

3. From the **Conversational DROs** group, set the parameters for the ID turning operation.

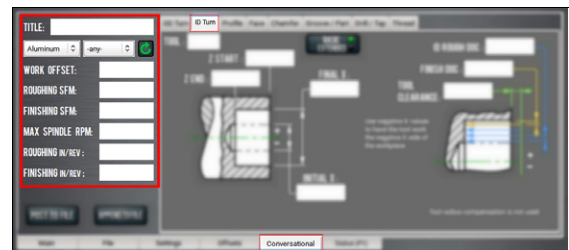


Figure 8-20: ID Turn tab on the Conversational tab.

4. Work through the program-specific DRO fields:
 - a. In the **Tool** DRO field, type the currently selected tool as it's defined in the **Tool Table** window (on the **Offsets** tab).
This DRO field is a command value — it sets the tool number for a tool change at the start of the program.
 - b. In the **Initial X** DRO field, type the diameter of the pilot hole. Make sure that the diameter is large enough to clear the tool holder's X width.
 - c. In the **Final X** DRO field, type the desired final diameter of the internal diameter. The value must be greater than twice the tool holder's X width plus tool clearance.
 - d. In the **Z Start** DRO field, type the location of the stock's face.

Note: This DRO field is used with the **Tool Clearance** DRO field to set the transition between rapid and feed rate on some Z moves.

- e. In the **Fillet Radius** DRO field, type the desired radius between the finished inside diameter and the face.

Note: The fillet calculation does not use CRC, so the middle of the fillet may not be on the true radius for a tool with a tip radius. Valid values are 0 or positive. Values larger than the Z range (Z START - Z END) or the X range $((\text{INITIAL X} - \text{FINAL X}) / 2)$ are valid, but will have a fillet start or end short of the finish locations, which may not be practical.

- f. In the **Z End** DRO field, type the desired final location for the part's face.
- g. In the **Tool Clearance** DRO field, type the distance required to retract the tool and transition between rapid and cutting feed rate. Because there's only one value used for X and Z moves, use the greater of the two clearances.

Note: Use larger values to begin; once you're familiar with how the program works, smaller values may save time. Larger values bring the back of the tool holder closer to the ID wall on the end of facing cuts.

- h. In the **ID Rough** DRO field, type the depth of material to cut on the radius of the bore. The depth of cut is adjusted to get the value used in the G-code.
- i. In the **Face Rough** DRO field, type the depth of material to cut on the internal face of the bore. The depth of cut is adjusted to get the value used in the G-code.

Note: The reverse or back cutting direction is sensitive to depth of cut. Form tools with a small angle between cutting edges allows for a larger depth of cut.

- j. In the **Finish DOC** DRO field, type the desired amount of material required for one finish pass on the ID, fillet, and face (completed after roughing).

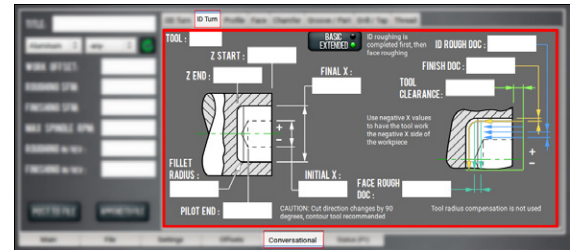


Figure 8-21: Program-specific DROs on the ID Turn tab.

About ID Turning

Internal diameter turning is the process of removing material from the inside of a part.

ID Turning in PathPilot

There are two versions of ID turning in PathPilot: basic and extended. Both versions use CSS for spindle speed control and FPR for feed control. The fillet does not use CRC so the fillet will not follow a true radius for tools with a tip radius.

• Basic Mode

Basic mode does one operation, which roughs and finishes from an initial pilot hole diameter to a final internal diameter without cutting a face at the bottom

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

of hole. Use Basic mode for through holes or holes that don't need a finished face. Each pass ends at Z End. Roughing starts at the pilot hole diameter (the value in the X Start DRO field), and incrementally cuts diameters with an adjusted depth of cut until the start of the finish diameter ($X\text{ End} - [2 \times \text{Finish DOC}]$). Finishing is done in one pass.

Extended Mode

Extended mode does three operations: ID roughing, face roughing, and an ID, fillet, and face finish pass. The extended ID roughing passes stop at the bottom of the pilot hole in order to prevent engaging too much of the tool cutting edge. Once the rough ID is cut to the pilot hole bottom, rough facing is started. There are two DRO fields for depth of cut, since, depending on the tool geometry, ID roughing and face roughing may need different depth of cuts.

For each face pass, the tool tip cuts to the hole center + Tool Clearance which requires a rough hole diameter (which was cut in the first operation) that is a little more than twice the tool's X width. Caution is needed to prevent hitting the back of the tool holder on the side of the hole.

Create a Profile on a Part

Using conversational programming, you can program PathPilot to rough and finish an arbitrary internal or external profile on a part. For information, see "About Profiling" (page 84).

Before You Begin

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

- The Tool Clear Dia X DRO field has a value of larger diameter than first X value on the Profile Point table.
- Verify that the X-axis direction matches your specific lathe.

X-Axis Differences Between Lathes



IMPORTANT! The X-axis direction differs between the 15L lathe and the 8L lathe. Where X-axis values are mentioned in this documentation, verify the intended movement direction based on your lathe model. Adjust clearances, cutting directions, and tool paths accordingly.

- **15L** X+ moves away from the operator, X- moves toward the operator.
- **8L** X+ moves toward the operator, X- moves away from the operator.

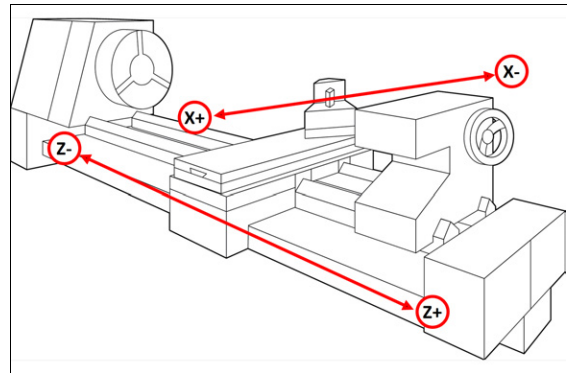


Figure 8-22: X-axis movement on an 8L lathe.

Complete the following steps in the order listed:

Describe the Stock

1. From the **Conversational** tab, select the **Profile** tab.
2. (Optional) To create an internal profile, select the button to toggle from External mode to Internal mode.
3. From the **Conversational DROs** group, set the parameters for the profiling operation.

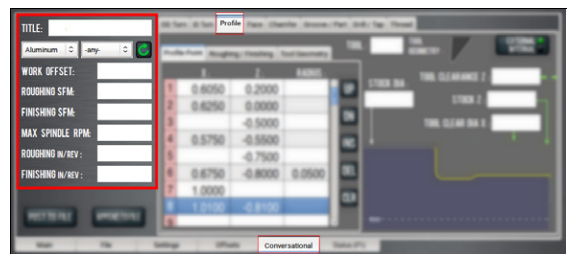


Figure 8-23: Conversational DROs on the Profile tab.

4. Work through the program-specific DRO fields:
 - a. In the **Stock Dia** DRO field, type the diameter of the stock.
 - b. In the **Tool Clearance Z** DRO field, type the incremental Z value for the tool clearance on the Z-axis.
Tool Clearance Z is the Z plane the tool goes from rapid to feed.
 - c. In the **Stock Z** DRO field, type the starting Z value for the profile.
 - d. In the **Tool Clear Dia X** DRO field, type the X value — as a diameter — for the tool clearance on the X-axis.
 - **For External Profiles** You must make sure the value typed in the **Tool Clear Dia X** DRO field is a smaller diameter than the value typed in the **Stock Dia** DRO field.
 - **For Internal Profiles** You must make sure the value typed in the **Tool Clear Dia X** DRO field is a larger diameter than the first diameter (specified in the **Profile Point** table).

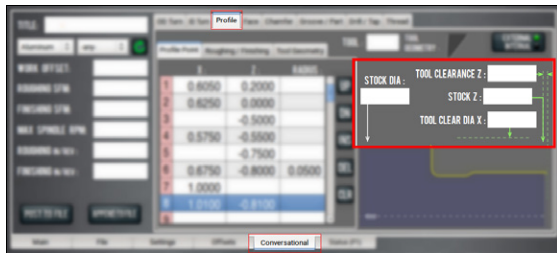


Figure 8-24: Program-specific DROs on the Profile tab.

Identify the Profile Points

Use the Profile Point table to describe the point-to-point values of a profile — from a larger Z value to a smaller Z value. As you work through the Profile Point table, PathPilot displays a graphical representation of the profile on your part. Click on any segment in the graphic to highlight the corresponding row in the Profile Point table. Alternately, you may click on any row in the Profile Point table to highlight the corresponding segment in the graphic. Point the mouse toward any area in the graphic and use the scroll wheel to zoom in and out to enlarge small features. To quit zooming, either select the Esc key or select another line in the Profile Point table.

To identify the profile points:

1. Select the **Profile Point** tab.
2. In the **Profile Point** table, type the X and Z values for the profile. X values are in diameters terms. Make sure that the X values are entered correctly based on your lathe model: for 8L lathes, the X+ values **decrease** toward the spindle.

Note: If the value is unchanged from the previous row in the **Profile Point** table, PathPilot assumes the value is repeated. If you are using the same value, you can leave the cell empty.
3. (Optional) In the **Radius** column, type a value to X and Z end points to create an arc.
 - a. **For a Center Point Above and/or to the Right of the Start Point** Type a positive radius value.
 - b. **For a Center Point Below and/or to the Left of the Start Point** Type a negative radius value.

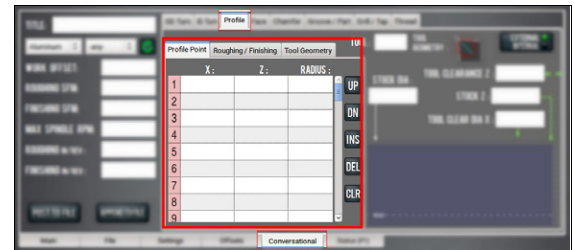


Figure 8-25: Profile Points table on the Profile tab.

Identify the Tool

1. In the **Tool** DRO field, type the number of the tool to use for creating the profile.
Tool Geometry displays a graphical representation of the selected tool.
2. On the graphical representation of the profile on your part, make sure there are no red line segments. If there are red line segments, you must specify a new tool or edit the fields in the **Profile Point** table.
A red line segment indicates that the geometry of the selected tool cannot cut the programmed angle without gouging the part profile — typically, when feature entry or exit angles are too steep for the tool geometry to clear.
The back angle of the tool will not clear the entry to the arc feature.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

Identify the Roughing and Finishing

Use the Roughing / Finishing tab to describe the required roughing and finishing for the part's profile.



Note: You can use only roughing or only finishing. It is an error if all three of the following DRO fields are empty: Finishing Passes, Roughing DOC, and Finishing DOC.

To identify the roughing and finishing:

1. Select the **Roughing / Finishing** tab.
PathPilot updates the graphical representation of the profile depending on which DRO field you select:
 - Select inside the **Roughing DOC** DRO field to display a roughing graphic.
 - Click inside the **Finishing DOC** DRO field or the **Finish Passes** DRO field to display a finishing graphic.
2. In the **Roughing DOC** DRO field, type the depth of cut for each roughing pass.



Note: The default value is 0.02 inches.

3. In the **Finishing DOC** DRO field, type the depth of cut for each finishing pass.



Note: The default value is 0.003 inches.

4. In the **Finishing Passes** DRO field, type the number – from 0-2 – of finishing passes.



Note: The default value is 2 (passes).

A finishing pass is a continuous pass from the start of the profile (toward the tailstock) to the end of the profile (toward the headstock).

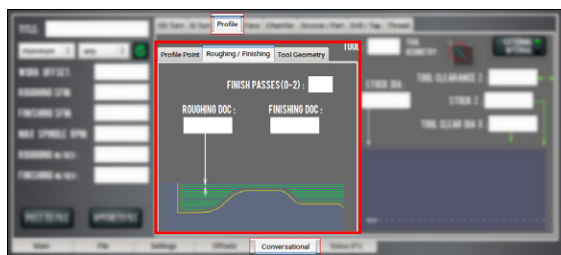


Figure 8-26: Roughing/Finishing tab on the Profile tab.

Describe the Tool Geometry

Use the Tool Geometry tab to describe the front and rear profiling angles. Because there are many tool and tool holder geometries, the Tool Geometry tab allows you to properly describe each tool and avoid part gouging.

To describe the tool geometry, first determine the cutting direction: either X+ or X-, based on the tool orientation and an internal or external profile. (For 8L lathes, an X+ cutting direction moves toward the operator.)

Use the table below to select the correct tool orientation for your machine.

Cutting Direction	Internal or External	Tool Orientation
X-	External	1, 2, 6
	Internal	3, 4, 8
X+	External	3, 4, 8
	Internal	1, 2, 6

To describe the tool geometry:

- **X- Cutting Tool** In the **Front Angle** DRO field and the **Back Angle** DRO field, type the value of the tool geometry expressed as a negative angle in the counterclockwise direction from 0°

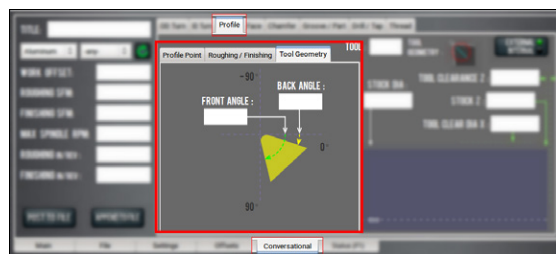


Figure 8-27: Tool Geometry tab on the Profile tab.



Note: The Tool Geometry graphic preview (to the right of the Tool DRO field) updates as angles are changed.

- **X+ Cutting Tool** In the **Front Angle** DRO field and the **Back Angle** DRO field, type the value of the tool geometry expressed as a negative angle in the clockwise direction from 0°

About Profiling

The profile is created from a list of points that describes the part geometry. A profile can have both forward (toward the tailstock) and rear (toward the headstock) facing features, and

can also start behind the highest Z plane (the feature that is closest to the tailstock).

On the Profile tab, you can also specify things like:

- Tool geometry
- Feeds and speeds
- Finish depth
- Number of finish passes
- Roughing depth of cut (roughing DOC)

Create a Face on a Part

Using conversational programming, you can program PathPilot to cut a face with tool paths from the stock's outer diameter to the spindle center or an inner diameter (with each pass progressing in Z toward the headstock). 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:

- To cut with a rear tool, the values used in the **Initial X** DRO field and the **Final X** must be positive. The tool works on the positive X side of the spindle center (the side toward you).
- To cut with a front tool, the values used in the **Initial X** DRO field and the **Final X** must be negative. The tool works on the negative side of the spindle (the side away from to you).
- The value used in the **Roughing DOC** DRO field must be positive.
- The value used in the **Finish DOC** DRO field must be positive.
- Spindle speed control: CSS.
- Feed rate control: FPR.

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.

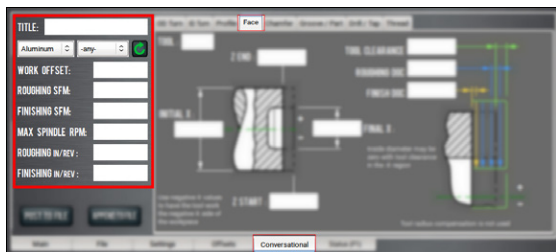


Figure 8-28: Conversational DROs on the Face tab.

3. Work through the program-specific DRO fields:
 - a. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
 - b. In the **Initial X** DRO field, type the stock diameter. This value is also used with the value in the **Tool Clearance** DRO field to locate some of the transitions between rapid and feed rate.
 - c. In the **Final X** DRO field, type the location of the face inside diameter. The tool path goes beyond this diameter by the tool clearance. For tools with a tip radius, the control point and face contact point aren't the same, so the tool clearance value, if greater than the tool tip radius, can be used to extend the path to the contact point.
 - d. In the **Z Start** DRO field, type the location of the stock face. Roughing passes start here. It is also used with Tool Clearance to set the transition between rapid and feed rates on some moves.
 - e. In the **Z End** DRO field, type the finished face location.
 - f. In the **Tool Clearance** DRO field, type the desired space required for tool retracting and transitions between rapid and cutting feed rate. Since there's one value used for X and Z moves, set the value to the greater of the two clearances. Larger values may be safer, but brings the back of the tool holder closer to the inner diameter wall on the end of facing cuts. Smaller values may save time once you're familiar with how well the program works.
 - g. In the **Roughing DOC** DRO field, type the depth of the material being cut. This depth of cut is adjusted to get the value used in the G-code.
 - h. In the **Finish DOC** DRO field, type the desired amount of material required for one finish pass (after roughing).

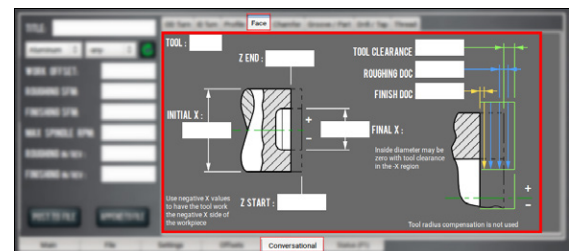


Figure 8-29: Program DROs on the Face tab.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

About Facing

During a facing routine, PathPilot does the following:

1. Rough facing starts at Z Start and incrementally cuts at the depth of cut until the start of the finish face pass (Z End + Finish DOC).
2. The start of each pass is at the Initial X diameter + Tool Clearance and moves in the minus X direction until the Final X diameter – Tool Clearance is reached.
If the value in the Final X DRO field is zero, the end of the pass will go beyond the spindle center.
3. Finishing is done in one pass at the value entered into the Finish DOC DRO field.

Create a Chamfer or Radius on a Part

Using conversational programming, you can program PathPilot to cut a chamfer, taper, or corner radius. For information, see "About Chamfer and Radius" (on the next page).

Before You Begin

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

- Uses cutter compensation (G41, G42), so that tools with a nose radius can cut to the correct profile.
- Radii are limited to 90° arcs that start on the outside diameter (the **Initial X** DRO field and the **Z End** DRO field). Be careful when using chamfer angles less than 30° or greater than 60°, due to the extra travel involved in traversing the tool clearance space at an angle. The path may take the tool into the chuck, spindle, or adjacent workpiece features.
- The value used in the **Roughing DOC** DRO field must be positive.
- The value used in the **Finish DOC** DRO field must be positive.
- Spindle speed control: CSS.
- Tool feed control: FPR.

To create a chamfer or radius on a part:

1. From the **Conversational** tab, select the **Chamfer** tab.
2. From the **Conversational DROs** group, set the parameters for the chamfer or radius operation.

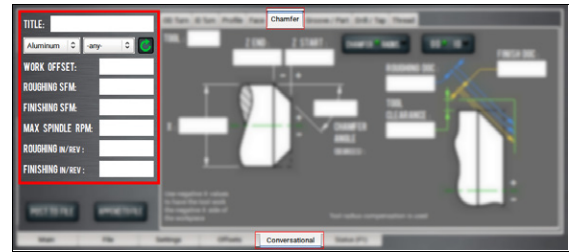


Figure 8-30: Conversational DROs on the Chamfer tab.

3. Work through the program-specific DRO fields:
 - a. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
 - b. In the **X** DRO field, type the stock diameter. This value is also used with the **Tool Clearance** DRO field to locate some of the transitions between rapid and feed rates.
 - c. In the **Z Start** DRO field, type the stock face or the end of the chamfer or radius. This value is also used with the Tool Clearance DRO field to set the transition between rapid and feed rates on some Z moves.
 - d. In the **Z End** DRO field, type the location of the start of the chamfer or radius. The Z width of a chamfer or the radius of a corner equals (Z End - Z Start).
 - e. (Optional) In the **Chamfer Angle** DRO field, type the angle between the workpiece centerline and the chamfer.
 - f. In the **Tool Clearance** DRO field, type the desired space beyond the stock outside diameter and face that's required for some movements to clear the workpiece. Since there is one value used for X and Z moves, set the value to the greater of the two clearances. Larger values may be safer; smaller values may save time once you're familiar with how well the program works. This field is also sometimes used as a location for retracting the tool while making cutting passes.
 - g. In the **Roughing DOC** DRO field, type the depth of cut during roughing. The depth of cut is adjusted. In this case, the roughing range is the distance from the workpiece corner (the intersection of the face and outer diameter) and the closest point on the chamfer or radius minus the finish depth of cut.
 - h. In the **Finish DOC** DRO field, type the desired amount of material required for one finish pass (after roughing).

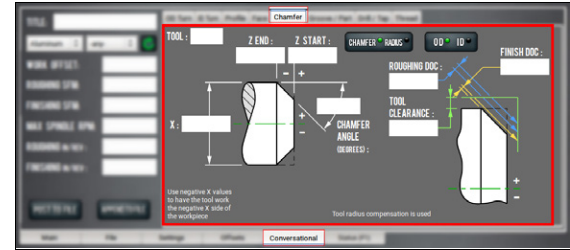


Figure 8-31: Program DROs on the Chamfer tab.

About Chamfer and Radius

During a routine to create a chamfer or a radius, PathPilot does the following:

1. Roughing starts at the corner of X and Z Start in adjusted depth of cut increments perpendicular to the chamfer angle or incremental arcs for radius.
2. The last roughing pass leaves enough material for the finish pass; finishing is done with a single pass.
3. Passes start and end on the perimeter of the tool clearance space, which is set by adding the tool clearance DRO value to the stock OD, X, and the face location, Z Start.

Create a Groove or Part a Workpiece

Using conversational programming, you can program PathPilot to create a groove or to part a workpiece from stock. For information, see "About Grooving and Parting" (page 89).

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

Before You Begin

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

- Grooving paths are based on **Z Start** and **Z End** values:
 - If the value in the **Z Start** DRO field is greater than the value in the **Z End** DRO field, the tool's control point is set to the +Z side of the tool.
 - If the value in the **Z Start** DRO field is less than the value in the **Z End** DRO field, the control point is set to the -Z side of the tool.
- Groove roughing is done with plunge cuts in the X direction. Each plunge is incremented in the Z direction from **Z Start ± Finish DOC** to **Z End ± (Tool Width + Finish DOC)**.
- Even though a grooving/parting tool may be considered to have two tips, valid tool orientation is limited to:
 - Groove on the positive side of Z Start, Front Tool = Type 3
 - Groove on the negative side of Z Start, Front Tool = Type 4
 - Part on the negative side of Z Start, Front Tool = Type 4
- CSS is used for spindle speed control.
- FPR is used for feed rate control.
- CRC is not used.

To create a groove on a part, or to part a workpiece:

1. From the **Conversational** tab, select the **Groove/Part** tab.
2. From the **Conversational DROs** group, set the parameters for the grooving or parting operation.

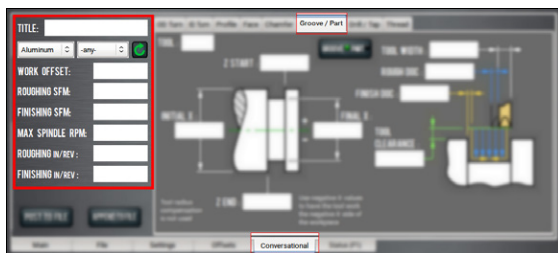


Figure 8-32: Conversational DROs on the Groove/Part tab.

3. If required, toggle the **Groove/Part** button. Then, do one of the following:
 - Go to "Create a Groove on a Part" (below).
 - Go to "Part a Workpiece from the Stock" (on the next page).

Create a Groove on a Part

Work through the program-specific DRO fields:

1. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
2. In the **Initial X** DRO field, type the stock diameter. This value is also used with the **Tool Clearance** DRO field to locate some of the transitions between rapid and feed rates.
3. In the **Final X** DRO field, type the diameter of the new groove bottom or the end of the parting.
4. In the **Z Start** DRO field, type the location of the groove start. For parting, this field sets the location of the +Z side of the slot.
5. In the **Z End** DRO field, type the location of the groove end.
6. In the **Tool Width** DRO field, type the groove or parting tool's width.
7. In the **Rough DOC** DRO field, type the depth of the material being cut. In this case, for groove, it is the Z offset for each plunge cut. The depth of cut is adjusted. Valid values are positive and normally should be less than the full depth width of the tool tip (usually the distance between tip radii centers).
8. In the **Finish DOC** DRO field, type the desired amount of material required for one finish pass (after roughing).
9. In the **Tool Clearance** DRO field, type the desired space beyond the stock outside diameter for rapid movements to clear the workpiece. Larger values may be safer; smaller values may save time once you're familiar with how well the program works. This field is also sometimes used as a location for retracting the tool between cutting passes.

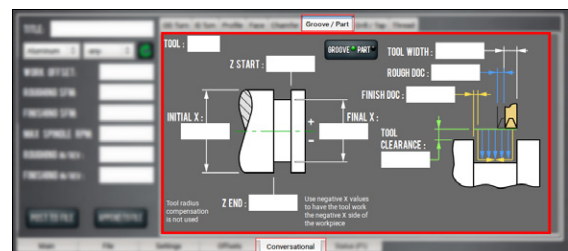


Figure 8-33: Program-specific DROs on the Groove/Part tab.

Part a Workpiece from the Stock

Work through the program-specific DRO fields:

1. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
2. In the **Initial X** DRO field, type the stock diameter. This value is also used with the **Tool Clearance** DRO field to locate some of the transitions between rapid and feed rates.
3. In the **Final X** DRO field, type the diameter of the new groove bottom or the end of the parting.
4. In the **Z Start** DRO field, type the location of the groove start. For parting, this field sets the location of the +Z side of the slot.
5. In the **Tool Width** DRO field, type the groove or parting tool's width.
6. In the **Tool Clearance** DRO field, type the desired space beyond the stock outside diameter for rapid movements to clear the workpiece. Larger values may be safer; smaller values may save time once you're familiar with how well the program works. This field is also sometimes used as a location for retracting the tool between cutting passes.

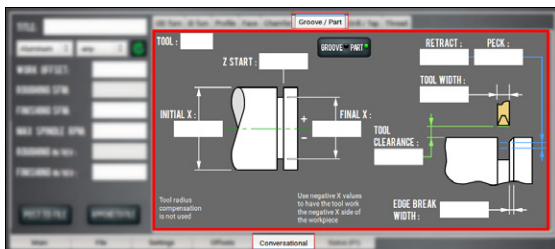


Figure 8-34: Program-specific DROs on the Groove/Part tab.

About Grooving and Parting

1. Groove finishing is done with a plunge cut down the Z End side.
2. When the tool reaches the bottom, the tool is moved in the Z direction toward the center of the groove, then retracts.
3. The tool is plunged on the Z Start side of the groove, then again is moved in Z toward the groove center and retracted. This requires a grooving tool, which can side cut. Part does one plunge cut at Z Start. The tool's

control point is on the +Z side of the tool. The plunge cannot be set to go beyond the spindle center (X = 0).

Create Holes on a Part or Tap a Hole

Using conversational programming, you can program PathPilot to drill holes on a part, or use rigid tapping to thread holes on a part. For information, see "About Drilling and Tapping" (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 **Z Start** DRO field must be larger than the value in the **Z End** DRO field.
- For tapping, the value used in the **Z End** DRO should allow for extra threading while the spindle comes to a stop and reverses.
- The value used in the **Peck Depth** DRO field needs a direction, so should have a negative value.
- Drilling is limited to the -Z direction, toward the spindle.
- Use the **RPM DRO** field instead of CSS.

To create holes on a part, or to tap a hole:

1. From the **Conversational** tab, select the **Drill/Tap** tab.
2. From the **Conversational DROs** group, set the parameters for the drilling or tapping operation.



Figure 8-35: Conversational DROs on the Drill/Tap tab.

3. If required, toggle the **Drill/Tap** button. Then, do one of the following:
 - Go to "Create a Hole on a Part" (below).
 - Go to "Create Threads in a Hole" (on the next page).

Create a Hole on a Part

Work through the program-specific DRO fields:

1. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

2. In the **Z Start** DRO field, type the stock face location. This field is also used with the **Tool Clearance** DRO field to set the transition between rapid and the feed for drilling or tapping.
3. In the **Z End** DRO field, type the final depth. This is the location where the drill feed stops and optionally dwells.
4. In the **Peck Depth** DRO field, type an incremental depth for retracting the drill to clear chips from the hole, if required. If drilling the hole doesn't need a peck, type 0. To make each peck depth equal, the value is adjusted to fit an integer number of pecks within the hole depth.
5. In the **Tool Clearance** DRO field, type the desired space required for tool retraction and transitions between rapid and cutting feed rate.
6. In the **Spindle RPM** DRO field, type the RPM (G97) desired.
7. In the **Dwell at Bottom (Sec)** DRO field, type the length of time that Z motion should pause so that the drill can finish cutting the hole bottom before retracting.

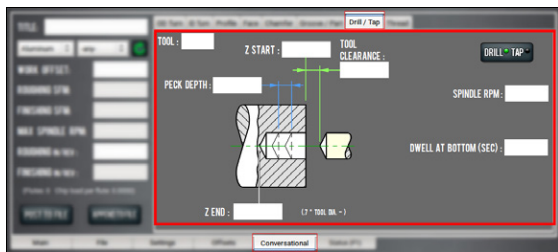


Figure 8-36: Program-specific DROs on the Drill/Tap tab.

Create Threads in a Hole

Work through the program-specific DRO fields:

1. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
2. In the **Z Start** DRO field, type the stock face location. This field is also used with the **Tool Clearance** DRO field to set the transition between rapid and the feed for drilling or tapping.
3. In the **Z End** DRO field, type the final depth. This is the location where the spindle rotation is reversed.
4. In the **Peck Depth** DRO field, type an incremental depth for retracting the drill to clear chips from the hole, if required. If drilling the hole doesn't need a peck, type 0. To make each peck depth equal, the value is adjusted to fit an integer number of pecks within the hole depth.
5. In the **Tool Clearance** DRO field, type the desired space required for tool retraction and transitions between rapid and cutting feed rate.
6. In the **Spindle RPM** DRO field, type the RPM (G97) desired.
7. In the **Threads/Inch (/mm)** DRO field, type the Z motion to spindle ratio that matches the thread pitch required. This field is also used with the **Pitch (Inches)** DRO field, so entering a value in either field calculates and inserts a value in the other.
8. In the **Pitch (Inches)** DRO field, type the Z motion to spindle ratio that matches the required thread pitch. This field is also used with the **Threads/Inch (/mm)** DRO field, so entering a value in either field calculates and inserts a value in the other.

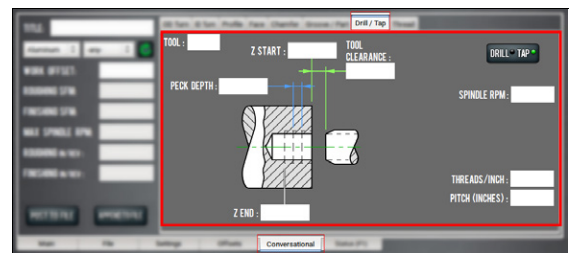


Figure 8-37: Program-specific DROs on the Drill/Tap tab.

About Drilling and Tapping

Drilling

1. For feed rate control, drill uses a millimeter or inch feed per revolution (G95) to feed from Z Start + Tool Clearance until Z End.
2. Rapids back to Z Start + Tool Clearance.

Dwell allows a pause for the drill to stay at Z End long enough to cut a full revolution at the bottom of the hole (rather than immediately retracting the drill, which could leave an irregular bottom).

Pecking can help clear chips before they bind in the hole during drilling. The peck motion retracts to Z Start + Tool Clearance on each cycle.



Note: Due to motion control limits, the retract to Z Start + Tool Clearance may not retract fully before starting the next drilling feed. You must verify that the pecking retract motion meets requirements for your application.

Tapping

Tap uses electronic gearing (G33.1, Rigid Tapping) to lock the Z-axis and spindle motion together for rigid tapping.

1. Tapping starts with the tap at Z Start + Tool Clearance.
2. Z motion waits until the spindle encoder index is tripped. Then, the gears are engaged and Z feeds at the rate set by the pitch or threads per unit (TPU) and spindle encoder count. The Z motion follows the spindle motion no matter what the spindle does.
3. For tapping, the spindle is run forward until Z End is reached, the spindle is reversed, which causes it to slow to a stop, then reverse. During this time, the tap continues to follow the spindle motion and continues to make threads until the spindle reverses.



Note: These extra threads needs to be considered when setting the Z End DRO field.

4. The reverse motion continues until reaching Z Start + Tool Clearance where the gearing is disengaged.

Create Threads on a Part

Using conversational programming, you can program PathPilot to single point an external or an internal thread on an existing outer diameter. For information, see "About Threading" (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 **Z Start** DRO field must be larger than the value used in the **Z End** DRO field.
- The value used in the **Depth of Cut** DRO field must be positive.

To create threads on a part:

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

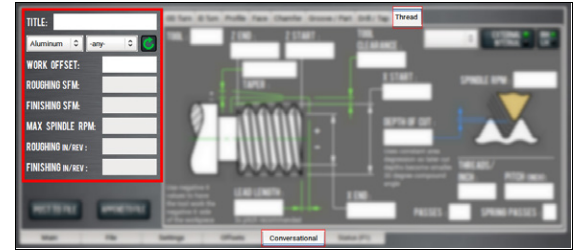


Figure 8-38: Conversational DROs on the Thread tab.

3. If required, toggle the **External/Internal** button or the **RH/LH** button.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

4. Work through the program-specific DRO fields:
 - a. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
 - b. In the **X Start** DRO field, type the existing major diameter for external threads, or minor diameter for internal threads.
 - c. In the **X End** DRO field, type the location of the new outside diameter.
 - d. In the **Z Start** DRO field, type the stock face location. This value is also used with the Tool Clearance DRO field to set the transition between rapid and feed rate on some Z moves.
 - e. In the **Z End** DRO field, type the final thread Z location.
 - f. In the **Tool Clearance** DRO field, type the desired space beyond the stock outside diameter that's required for rapid movements to clear the workpiece. This sets the starting X diameter of the thread cycle return path. Larger values may be safer; smaller values may save time once you're familiar with how well the program works.
 - g. In the **Lead Length** DRO field, type the required length during the start of the cutting path that allows the motion to stabilize before cutting material. The start of a cutting pass waits for the spindle encoder index to trip; when it does, the Z motion tries to instantly match the spindle speed, but actually needs time to accelerate and match the spindle encoder count. This value must be a compromise between spindle speed, thread pitch, and workpiece clearance.
 - h. From the **Thread Table** drop-down menu, select the value for the thread. The threads listed in this menu follow the current unit setting (inch or millimeter). Once you make a selection, the data for that thread is entered into the appropriate DRO fields.
 - i. In the **Spindle RPM** DRO field, type the spindle rpm.
 - j. In the **Depth of Cut** DRO field, type the depth of material being cut. Each pass is incremented based on a calculation of the area of the material being removed. This allows for a constant tool load for each pass.
 - k. In the **Threads/Inch (/mm)** DRO field, type the number of threads per inch (or millimeter).
 - l. In the **Pitch (Inch)** DRO field, type the distance per thread.
 - m. In the **Passes** DRO field, type the number of successive passes to cut the thread.
 - n. In the **Spring Passes** DRO field, type the number of extra passes at full thread depth. We suggest using this to clean up the thread and compensate for any material deflection during thread cutting.

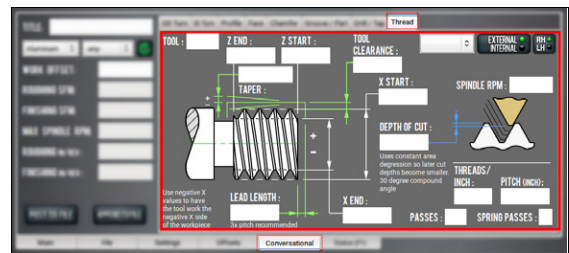


Figure 8-39: Program-specific DROs on the Thread tab.

About Threading

Thread is based on the G76 Threading Cycle. This canned cycle contains a lead-in, cut, lead out, and return path for each threading pass. Each cycle is incrementally offset in X and Z to account for a 30° software compound angle. The offset is calculated such that the each pass cuts the same amount of material by cross sectional area. The first pass has the most X displacement, and this decreases with each pass.

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

8.2 MACHINE SETTINGS AND ACCESSORIES

Before running a G-code program, you must first make sure that the machine settings are properly configured.

8.2.1 Enable an Internet Connection	93
8.2.2 Change the Network Name	94
8.2.3 Change the Screen Orientation	94
8.2.4 Specify the Spindle Pulley Option	95
8.2.5 Specify the Tool Change Method	96
8.2.6 Disable Limit Switches	96
8.2.7 Limit G30 Moves	97
8.2.8 Enable the On-Screen Keyboard	97
8.2.9 Enable Tooltips	97
8.2.10 Enable the USB M-Code I/O Interface Kit	98
8.2.11 Enable Feeds and Speeds Suggestions in Conversational Routines	98
8.2.12 Use a USB Camera	98

8.2.1 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 PathPilot HUB instead of a USB drive.

To enable an internet connection:

1. From the PathPilot interface, on the **Status** tab, select **Internet**.



Figure 8-40: Internet button on the Settings tab.

The **Network Configuration** dialog box displays.

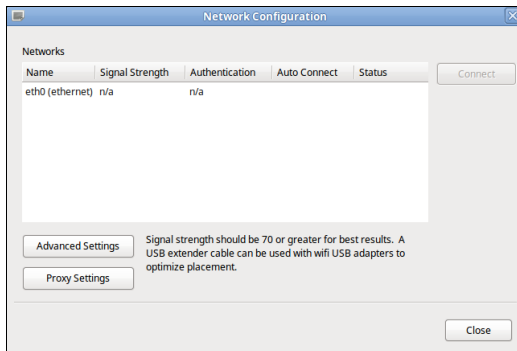


Figure 8-41: Network Configuration dialog box.

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.

Enable Automatic Updates

Note: Automatic updates require an internet connection. If you haven't yet enabled it, go to "Enable an Internet Connection" (above).

If desired, you can enable automatic updates for PathPilot. To enable automatic updates:

1. From the PathPilot interface, on the **Status** tab, select **Update**.

The **Software Update** dialog box displays.

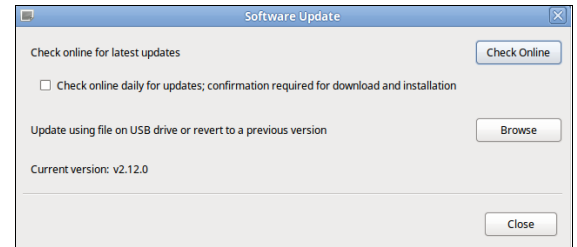


Figure 8-42: Software Update dialog box.

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.

8: PATHPILOT TOOLS AND FEATURES

8.2 Machine Settings and Accessories

8.2.2 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.

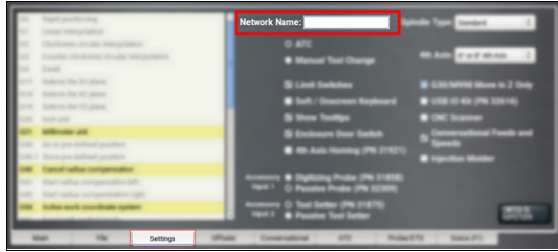


Figure 8-43: Network Name field on the Settings tab.



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.

8.2.3 Change the Screen Orientation

A vertical orientation for 1920 × 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.

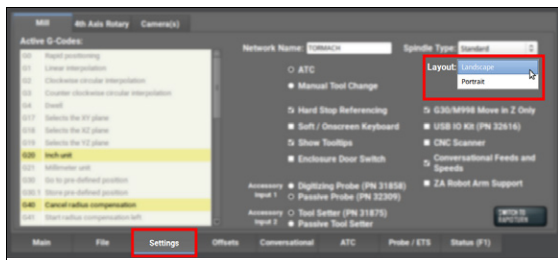


Figure 8-44: Layout drop-down menu on the Settings tab.

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.

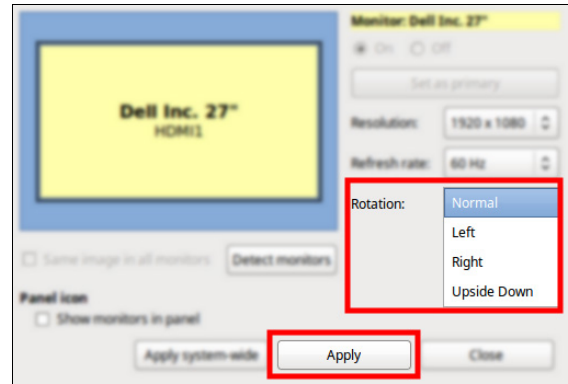


Figure 8-45: Monitor configuration dialog box.

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.

8: PATHPILOT TOOLS AND FEATURES

8.2 Machine Settings and Accessories

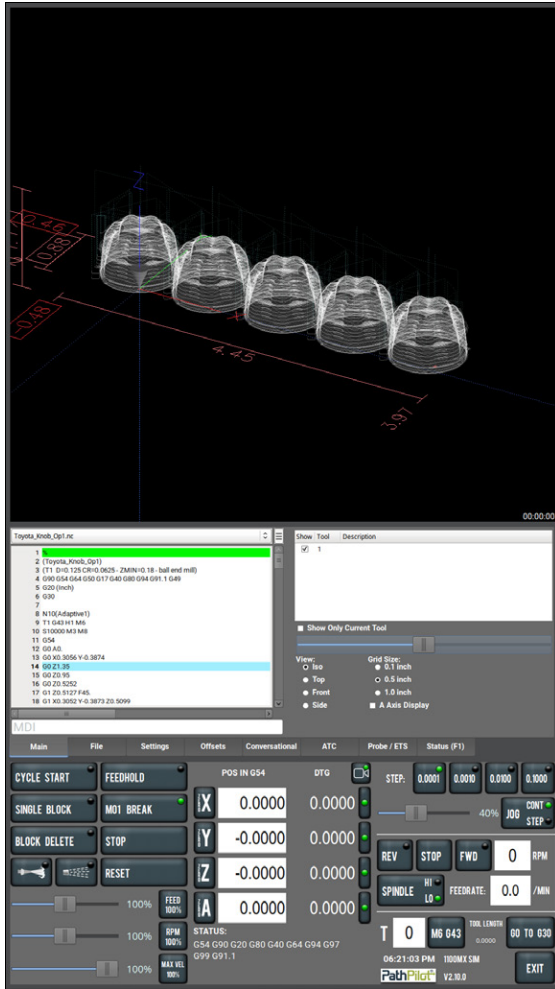


Figure 8-46: Tool Path window in portrait screen layout.

- 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.

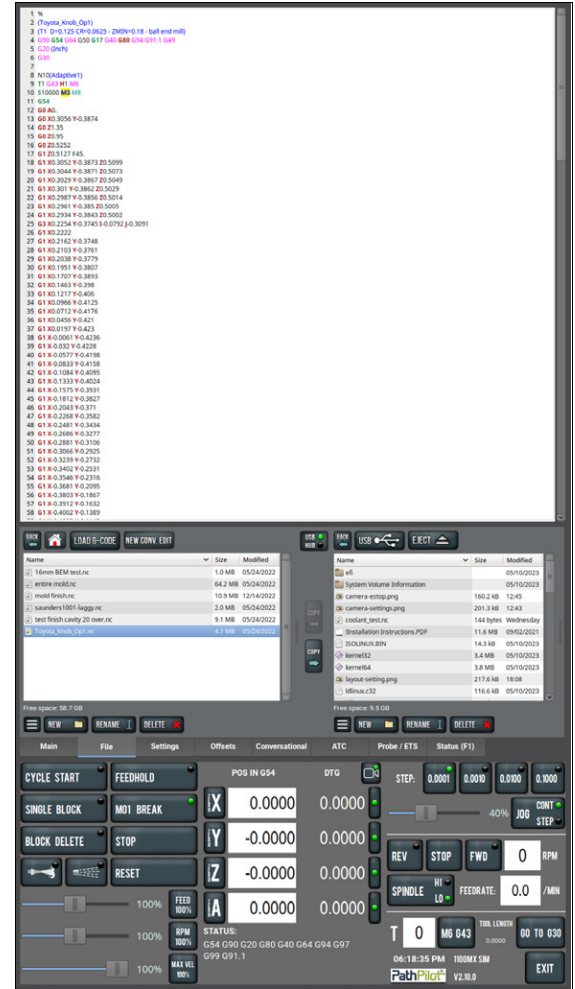


Figure 8-47: File tab G-code preview in portrait screen layout.

8.2.4 Specify the Spindle Pulley Option

- From the Settings tab, select one of the following:
 - Collet (High Speed)
 - Chuck (Low Speed)

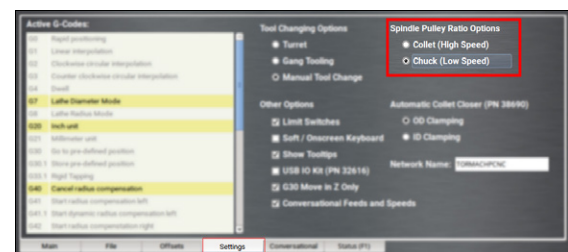


Figure 8-48: Spindle pulley options on the Settings tab.

8: PATHPILOT TOOLS AND FEATURES

8.2 Machine Settings and Accessories

8.2.5 Specify the Tool Change Method

When PathPilot finds a T 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.

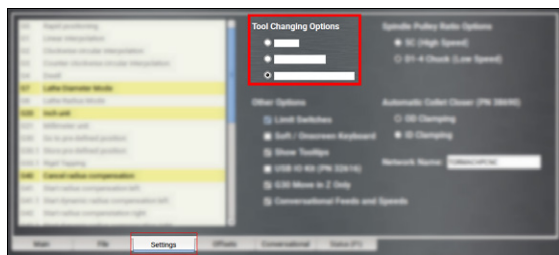


Figure 8-49: Tool change method options on the Settings tab.

About Gang Tool Changes

If the gang option is selected, the T command simply applies the offsets for the selected tool. Unlike the manual tool change option, T commands with the gang option selected doesn't pause the program during the tool change. Offsets are applied nearly instantaneously, and machining resumes automatically during part program execution.

About Manual Tool Changes

The manual tool change option causes the machine to pause at the T command during G-code program execution. This allows you to manually change tools on a quick-change tool post. After changing tools, selecting Cycle Start resumes program execution with the new tool offsets applied. When the machine is paused waiting for a manual tool change in the middle of a G-code program, the light on the Cycle Start button flashes, and the tool label flashes with the requested tool number.

About Mixed Tooling Changes

When mixing turret or gang tools with quick-change tool post mounted tools, there are two choices:

1. If you select the turret or gang setting, the machine won't automatically pause for manual changes of quick-change tool post tools. You must program these manual M01 stops.



Note: We recommend this option.

2. If you select the quick-change tool post option, you must select Cycle Start to confirm both manual and turret/gang changes.

8.2.6 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" (below).



Note: By default, the Limit Switches checkbox is selected.

To disable limit switches:

1. From the **Settings** tab, clear the **Limit Switches** checkbox.

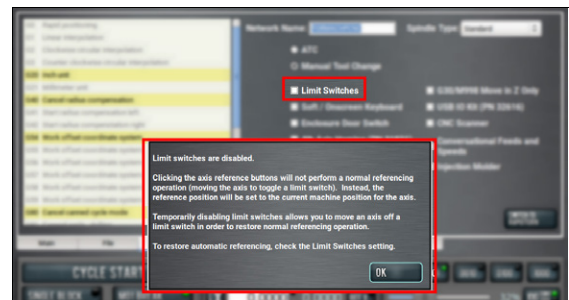


Figure 8-50: Limit Switches checkbox on the Status tab.

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.

About Limit Switches

In the PathPilot interface, on the Settings tab, the Limit Switches checkbox is selected by default.

If the checkbox is cleared, the machine completes a unique referencing procedure after selecting Ref X and Ref Z: rather than moving each axis to the end of its travel, the reference position is set as the machine's current position. This is useful for troubleshooting: if the limit switches are disabled, you're able to move the axis off of its limit switch.

8.2.7 Limit G30 Moves

You can limit G30 moves so that only the Z-axis moves. For information, see "About G30" (page 115).

To limit G30 moves:

- From the **Settings** tab, select **G30 Move in Z Only**.

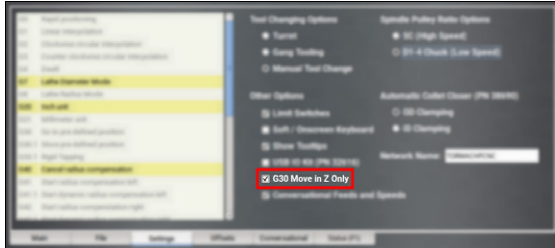


Figure 8-51: Settings tab.

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 114).

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.

8.2.8 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**.

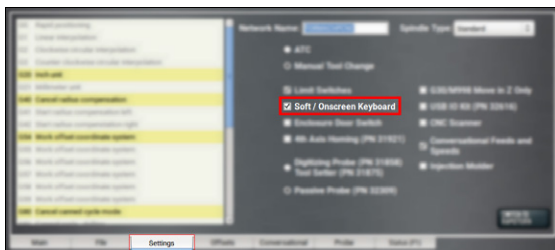


Figure 8-52: Settings tab.

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 8-53: Soft (on-screen) keyboard.

8.2.9 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:

8: PATHPILOT TOOLS AND FEATURES

8.2 Machine Settings and Accessories

1. From the Settings tab, select or clear **Show Tooltips**.

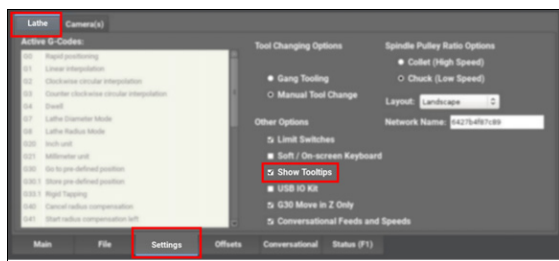


Figure 8-54: Show Tooltips checkbox.

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.

8.2.10 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)**.

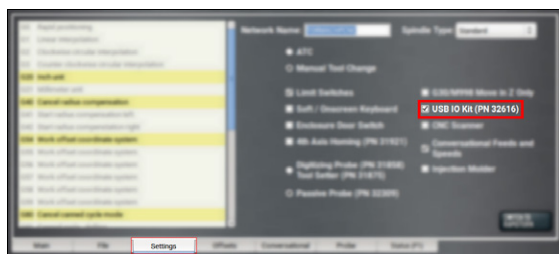


Figure 8-55: Settings tab.

8.2.11 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 116).

- From the **Settings** tab, select **Conversational Feeds and Speeds**.

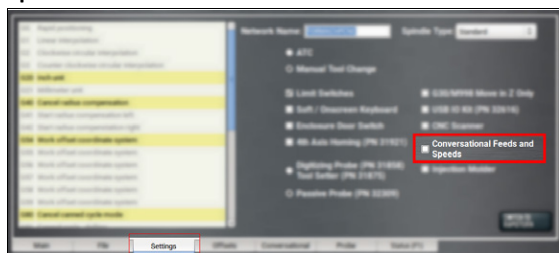


Figure 8-56: Settings tab.

8.2.12 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 8-57: 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.

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 8-58: Manual recording controls.

- Select the **Video Camera Recording** button in the **Persistent Controls** section.

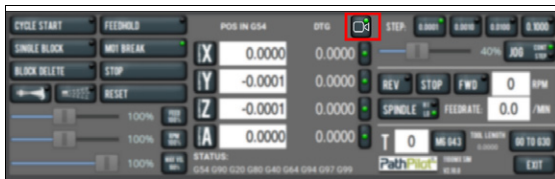


Figure 8-59: Video Camera Recording button.

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.

To include a screen recording:

1. Toggle the **Include PathPilot screen in recordings** checkbox in the **Camera Settings** area of the **Camera(s)**

tab to enable or disable screen recording.



Figure 8-60: 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.

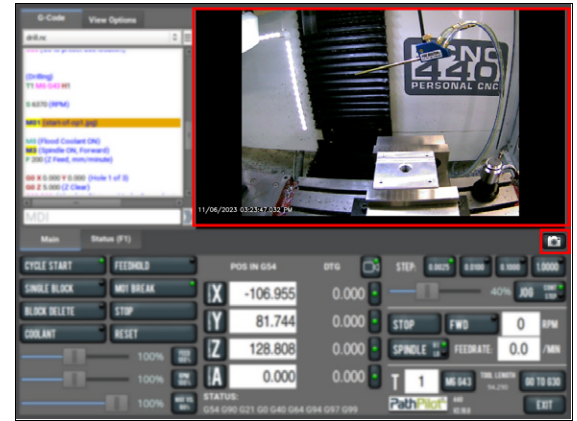


Figure 8-61: Example of taking a photo.

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.

8: PATHPILOT TOOLS AND FEATURES

8.2 Machine Settings and Accessories

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:

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 159).

8: PATHPILOT TOOLS AND FEATURES

8.3 Set Up G-Code Programs

8.3 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.

- 8.3.1 Set Tool Geometry Offsets..... 102
- 8.3.2 Touch Off the Tool Geometry Offsets..... 102
- 8.3.3 Set Work Offsets.....103
- 8.3.4 View Work Offsets.....104
- 8.3.5 View Available G-Code Modes..... 104

8.3.1 Set Tool Geometry Offsets

Before running a G-code program, PathPilot must know the geometry of the tools that are required for the program. For more information on using tool length offsets, see "About Tool Offsets" (page 136).

Note: You can import a .csv file with tool geometry offset data. For information, see "Import and Export the Tool Table" (page 137).

To set tool geometry offsets:

1. Verify that the machine is powered on, out of reset, and the axes have been referenced.
2. From the PathPilot controller, on the **Offsets** tab, select the **Offsets Table** tab.
3. Find the **Tool Table** window.



Figure 8-62: Tool Table window on the Offsets tab.

4. Touch off the tool geometry offsets. For information, see "Touch Off the Tool Geometry Offsets" (page 136).
5. (Optional) Select a field to edit. When finished, select the **Enter** key.

8.3.2 Touch Off the Tool Geometry Offsets

On the Offsets tab, you can use the Tool Touch tab to graphically select a tool, and then touch off the tool to set the geometry offsets.

Tip! When using this method to measure your tool geometry, remember that the X zero location never changes (the spindle centerline is always X = 0), but the Z zero location may change depending on the length of the workpiece that is chucked into the spindle. As long as each tool is measured to a face that has been zeroed, only measure these tools one time or until you replace an insert.

To touch off the tool geometry offsets:

- From the **Offsets** tab, on the **Tool Touch** tab, select a tool.
The tools along the bottom of the screen are front tool post tools (used by machines with a quick-change tool post setup).

Note: Gang tooling setups typically use both front and rear tool post tools.

After you select the tool, PathPilot:

- Sets the tip orientation for the tool, used (along with tip radius) in cutter compensation.
- Sets the tool type (front tool post or rear tool post), used by the conversational routines to double check the user entry fields in an attempt to try to detect and prevent crashes.
- Displays the tool touch off dialog.

Touch X

1. Take a skim cut off of the diameter of the workpiece — just long enough to measure the cut surface with a micrometer.
2. Jog the tool away from workpiece in Z, but don't jog the machine in X.
3. Measure the diameter of the skim-cut workpiece with a micrometer.
4. In the **Touch X** DRO field, type the value that you measured in Step 3.

Note: If you're touching off a front tool post tool, verify that the value you enter is positive.

5. Select **Touch X**.
The LED comes on.

Touch Z

1. Jog the machine toward the part zero (usually the face of the workpiece) in the Z direction.
2. Move the tool so that its cutting edge is just touching the surface of the material and define this as Z = 0. Use a sheet of paper to indicate when the tool is touching the material.
3. Slowly jog the Z-axis until it's approximately 1/4 in. away from part zero on the workpiece.
4. With the paper between the tool and the workpiece, slowly jog the machine until you feel a light pull on the paper.
5. In the Touch Z DRO field, type the thickness of the piece of paper. Then select **Touch Z**.

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 gang tooling, different X/Z positions on the carriage).

Tool offsets are broken down into two components:

- **Geometry Offsets** Represents the distance from the work offset zero location to the tool's control point.



Note: Unlike on a mill (where G43 must be called out to apply an offset), tool geometry offsets are automatically applied with the Txx tool change command.

- **Wear Offsets**

The sign convention for the machine are as follows:

- Z negative is toward the spindle.
- X positive is toward the operator.

Sign convention is important when you choose the manual tool change option or the gang tooling option.

All tools mounted for use on the operator side of the workpiece are touched off using positive X (diameter) values, and most X words in part programming for these tools have positive values.

8.3.3 Set Work Offsets

In the the **X (Dia)** DRO field, all values are expressed in terms of the diameter of the workpiece, not the radius.



EXAMPLE

If the active tool is 2 in. away from the spindle centerline, **4.000** displays in the **X DRO** field.

To set the current axis location to zero in the active work coordinate system:

- Select **Zero**.

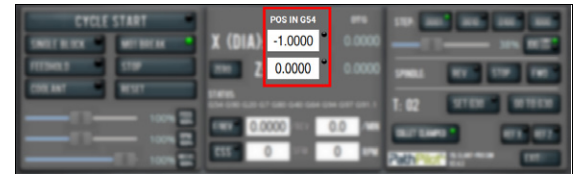


Figure 8-63: Work Offset DRO fields.

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 8-64: 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 134).

About Work Offsets

Work offsets allow you to think in terms of X 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.

8.4 RUN G-CODE PROGRAMS

While running a G-code program, use the following controls:

8.4.1 Bring the Machine Out of Reset	105
8.4.2 View the Active Axis to Jog	105
8.4.3 Jog the Machine	105
8.4.4 View the Current Machine Position	106
8.4.5 Reference the Machine	106
8.4.6 Start a Program	107
8.4.7 Stop Machine Motion	107
8.4.8 Operate the Coolant Pump	107
8.4.9 View the Active G-Code Modes	108
8.4.10 View the Distance to Go	108

8.4.1 Bring the Machine Out of Reset

- Select **Reset**.



Figure 8-67: Reset button.

For more information on reset mode, see "About Reset Mode" (page 127).

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.

8.4.2 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.

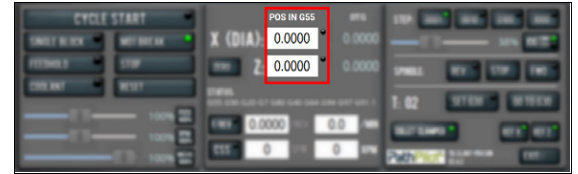


Figure 8-68: Work Offset DRO fields.

For information, see "Jog the Machine" (page 129).

8.4.3 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 8-69: 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.

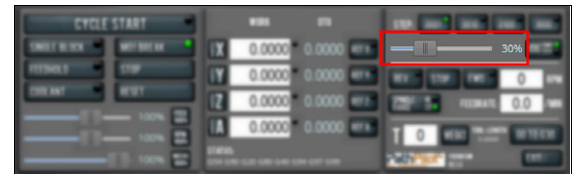


Figure 8-70: Jog Speed slider.

For more information on continuous velocity mode, see "About Continuous Velocity Jogging" (page 129).

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 8-71: Step buttons (in G20 mode).

8: PATHPILOT TOOLS AND FEATURES

8.4 Run G-Code Programs

- 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 129).

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**.

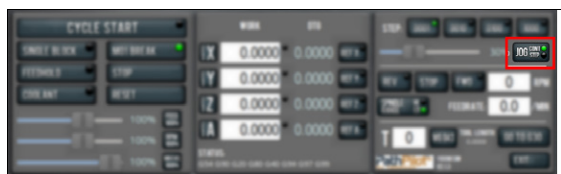


Figure 8-72: Continuous velocity jogging controls.

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 8-73: Jog Speed slider.

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
- The Operator Console's pendant wheel is turned in the desired direction

This is useful when you're doing things like:

- Roughly positioning the machine (for example, to move the cutting tool away from the workpiece).
- Moving the machine a certain distance at a constant speed (for example, to manually face off a workpiece).

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 8-74: Step buttons (in G20 mode).

About Step Jogging

While jogging in step mode, the machine moves one step at a time. 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 offsets.

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. The Operator Console's pendant moves the machine in the step size that's selected with the three-position switch.

8.4.4 View the Current Machine Position

- Identify the **Work Offset** DRO fields.

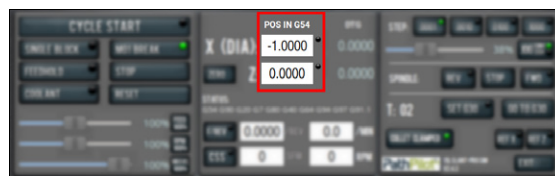


Figure 8-75: Work Offset DRO fields.

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 134).

8.4.5 Reference the Machine

1. Verify that the machine can freely move to its reference position (at the ends of travel).

- 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 8-76: Reference buttons.

- 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 128).

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.

8.4.6 Start a Program

- From the PathPilot interface, in the **Main** tab, select **Cycle Start**.

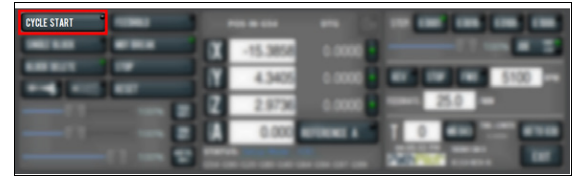


Figure 8-77: Cycle Start button.

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. Selecting Cycle Start causes an alarm unless the chuck guard or enclosure door is closed and a valid G-code program is loaded. 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 63).
- Before you've loaded a G-code program. For information, see "Load G-Code" (page 133).
- Before referencing the machine. For information, see "Reference the Machine" (page 128).

8.4.7 Stop Machine Motion

- From the **Program Control** area, select **Stop**.

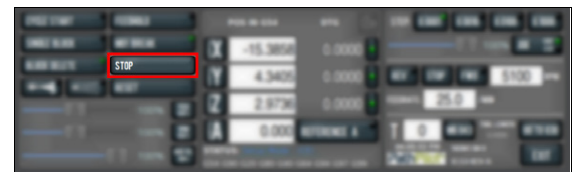


Figure 8-78: Stop button.

8.4.8 Operate the Coolant Pump

To turn coolant on or off:

8: PATHPILOT TOOLS AND FEATURES

8.4 Run G-Code Programs

- Select **Coolant**.

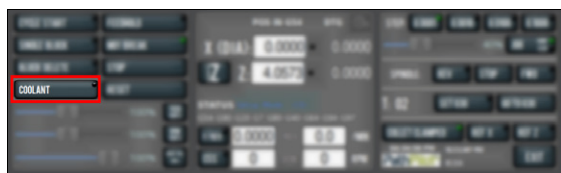


Figure 8-79: Coolant button.

For more information on turning on and off coolant, see "About Coolant" (page 138).

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.

8.4.9 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 8-80: Status read-only DRO field.

For more information on G-code modes, go to "View Available G-Code Modes" (page 104).

8.4.10 View the Distance to Go

To view the distance to go:

- Identify the **DTG** read-only DRO fields.

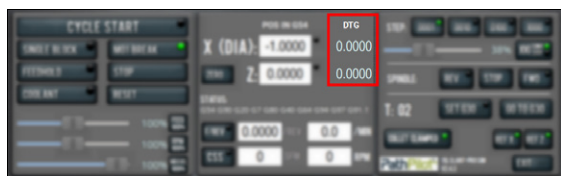


Figure 8-81: DTG read-only DRO fields.

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.

8.5 CONTROL G-CODE PROGRAMS

If necessary, use the following controls to add to your G-code program:

8.5.1 Use the Feed Hold Function	109
8.5.2 Use the Feed Rate Override Function	109
8.5.3 Use M01 Break Mode	110
8.5.4 Use the Maxvel Override Function	110
8.5.5 Use Single Block Mode	110
8.5.6 Use the Spindle Override Function	110
8.5.7 Change the Feed Rate	111
8.5.8 Change the Spindle Speed	111
8.5.9 Change the Feed Rate Mode	112
8.5.10 Change the Spindle Speed Command Mode	113
8.5.11 Change the Tool Number	114
8.5.12 Use a G30 Position	114
8.5.13 Manually Enter Commands	115
8.5.14 Copy Recently Entered Commands	116
8.5.15 Use Feeds and Speeds Suggestions	116
8.5.16 Use Cycle Counters (M30 and M99)	119

8.5.1 Use the Feed Hold Function

- Select **Feed Hold**.

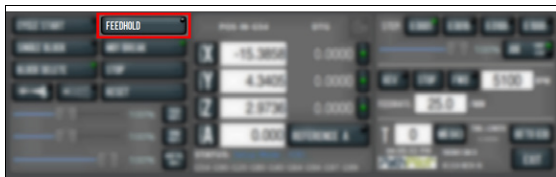


Figure 8-82: Feed Hold button.

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 107).

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.

8.5.2 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 8-83: 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.

8: PATHPILOT TOOLS AND FEATURES

8.5 Control G-Code Programs

8.5.3 Use M01 Break Mode

- Select **M01 Break**.



Figure 8-84: 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 107).
- **When M01 Break is Inactive** PathPilot ignores all programmed M01 commands.

8.5.4 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 8-85: 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%.

8.5.5 Use Single Block Mode

- Select **Single Block**.

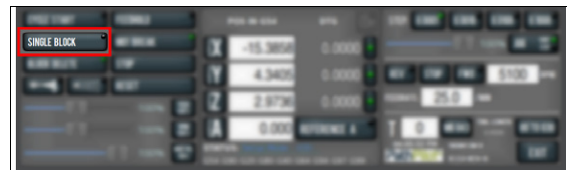


Figure 8-86: Single Block button.

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 107).

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.

8.5.6 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.

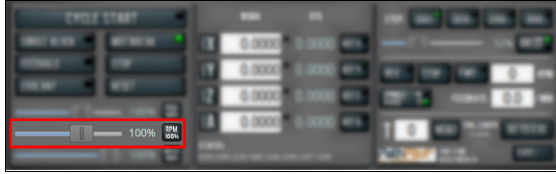


Figure 8-87: Spindle Override slider.

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%.

8.5.7 Change the Feed Rate

- In the **Feed Rate** DRO field, type in a feed rate. Then select the **Enter** key.



Figure 8-88: Feed Rate DRO field.

For information, see "About Feed Rates" (below).

About Feed Rates

There are two feed rate modes:

- **Units per Minute (G94)** A G-code F word is interpreted to mean the controlled point should move at a certain number of inches or millimeters per minute, depending upon what length units are being used and which axis or axes are moving.
- **Units per Revolution of Spindle (G95)** A G-code F word is interpreted to mean the controlled point should move at a certain number of inches or millimeters per spindle revolution depending upon what length units are being used and which axis or axes are moving.

Note: We recommend programming in units per revolution for almost all machining done on a lathe.

These settings are mutually exclusive: you cannot simultaneously be in both units per revolution and units per minute modes. The F/Min and F/Rev DRO fields display the current F word in both units/revolution and units/minute. The active mode is indicated in the PathPilot interface: when units per revolution mode is active, the light on the F/Rev button illuminates, the /Rev label (next to the DRO field) is white, and the Min label is gray.

You can turn on units per revolution mode by:

- Clicking in the F/Rev DRO field, entering a value, and pressing the Enter key on the keyboard.
- Clicking the F/Rev button.
- Typing G95 into the MDI Line DRO field, and pressing the Enter key on the keyboard.
- Running a G-code program with a G95 command.

8.5.8 Change the Spindle Speed

- In the **Spindle RPM** DRO field, type in a spindle speed. Then select the **Enter** key.

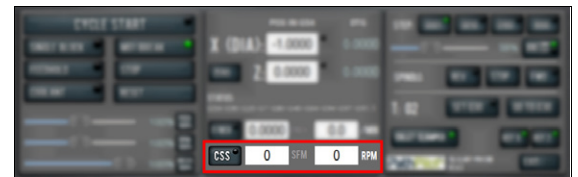


Figure 8-89: Spindle RPM DRO field.

For information, see "About Spindle Controls" (page 113).

8: PATHPILOT TOOLS AND FEATURES

8.5 Control G-Code Programs

About Spindle Controls

A spindle speed is the rate at which the spindle rotates.

Button	G-Code	Use to...
FWD	M03	In G97 mode: start the spindle clockwise at the RPM specified in the RPM DRO field. In G96 mode: start the spindle at an RPM specified by the SFM DRO field and the X (Dia.) value in the X DRO field.
REV	M04	In G97 mode: start the spindle in the reverse direction at the RPM specified in the spindle RPM DRO field. In G96 mode: start the spindle at an RPM specified by the SFM DRO field and the X (Dia.) value in the X DRO field.
Stop	M05	Stop the spindle.

The FWD and REV buttons and the RPM DRO field don't operate if they're selected when:

- A G-code program is running.
- Using manual data input (MDI) commands.

There are two spindle speed command modes:

- **Constant RPM (G97)** The spindle turns at an rpm equal to the S word.

EXAMPLE

G97 S500 M03 starts the spindle at 500 rpm.

- **Constant Surface Speed (G96)** The spindle rpm varies to maintain a constant speed in terms of linear feet per minute over the workpiece. The actual spindle RPM depends on the diameter of the work being turned, increasing as diameters get smaller and decreasing as diameters get larger.

EXAMPLE

G96 S50 M03 turns the spindle on, and the linear speed at which the cutting tool's control point travels over the work piece's surface is 50 feet per minute.



Note: Because tool life and efficiency depends on surface speed (and not rpm), we recommend constant surface speed mode for all machining on a lathe.

8.5.9 Change the Feed Rate Mode

To turn on F/Rev mode, do any of the following:

- Select the **F/Rev** DRO field and enter a value. Then select the **Enter** key.



Figure 8-90: A value in the F/Rev DRO field, which turns on F/Rev mode.

The button's light turns on, and the F/Min DRO field's label turns gray.

- Select **F/Rev** when its light is off.
- Select the **MDI Line** DRO field and type G95. Then select the **Enter** key.
- Run a G-code program that has a G95 command.

You can turn off F/Rev mode by doing any of the following:

- Select the **F/Min** DRO field and enter a value. Then select the **Enter** key.



Figure 8-91: A value in the F/Min DRO field, which turns off F/Rev mode.

The F/Rev button's light turns off, and the F/Rev DRO field's label turns gray.

- Select **F/Rev** when its light is off.
- Select the **MDI Line** DRO field and type G94. Then select the **Enter** key.
- Run a G-code program that has a G94 command.

For more information on feed rate modes, see "About Feed Rate Modes" (below).

About Feed Rate Modes

The machine recognizes two feed rate modes: units per minute (F/Min) or units per revolution (F/Rev). These settings

are mutually exclusive — you can't simultaneously be in both F/Rev and F/Min modes. The F/Min and F/Rev DRO fields display the current F word in both units per revolution and units per minute, with the active mode indicated by both the F/Rev button's light and the DRO fields' labels. For example, when F/Rev is active, the F/Rev button light is on, the F/Rev label is white, and the F/Min label is gray.

Units per Minute Mode (G94)

In units per minute feed rate mode, a G-code F word is interpreted to mean the controlled point should move at a certain number of inches or millimeters per minute, depending upon what length units are being used and which axis or axes are moving.

Units per Revolution Mode (G95)

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



Note: For almost all machining on a lathe, programming in units per revolution is more desirable.

8.5.10 Change the Spindle Speed Command Mode

To turn on constant surface speed mode (G96):

- Select the **SFM** DRO field and enter a value. Then select the **Enter** key.

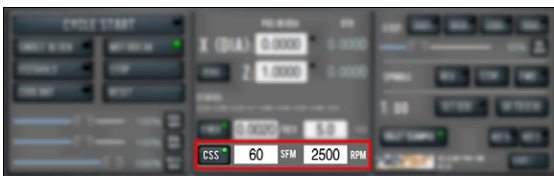


Figure 8-92: SFM DRO field.

To turn on constant RPM mode (G97):

- Select the **RPM** DRO field and enter a value. Then select the **Enter** key.

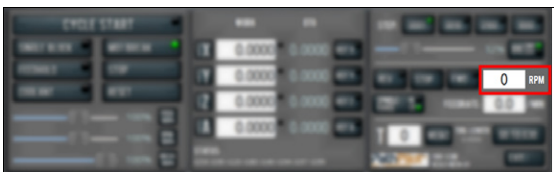


Figure 8-93: RPM DRO field.

For more information on spindle speed command modes, see "About Spindle Speed Command Modes" (below).

About Spindle Speed Command Modes

The machine recognizes two spindle speed command modes: constant RPM (G97) and constant surface speed (G96).

Constant Surface Speed Mode (G96)

In constant surface speed mode (G96), the spindle RPM varies to maintain a constant speed in terms of linear feet per minute over the workpiece.



EXAMPLE

G96 S50 M03 turns the spindle on, and the linear speed at which the cutting tool's control point travels over the work piece's surface is 50 feet per minute.

In constant surface speed mode, the actual spindle RPM depends on the diameter of the work being turned: it increases as diameters get smaller, and decreases as diameters get larger.



Tip! Because tool life and efficiency depends on surface speed (and not RPMs), we recommend operating in constant surface speed mode.

Constant RPM Mode (G97)

In constant RPM mode (G97), the spindle turns at an RPM equal to the S word.



EXAMPLE

G97 S500 M03 starts the spindle at 500 RPMs

About Spindle Controls

A spindle speed is the rate at which the spindle rotates.

Button	G-Code	Use to...
FWD	M03	In G97 mode: start the spindle clockwise at the RPM specified in the RPM DRO field. In G96 mode: start the spindle at an RPM specified by the SFM DRO field and the X (Dia.) value in the X DRO field.

8: PATHPILOT TOOLS AND FEATURES

8.5 Control G-Code Programs

Button	G-Code	Use to...
REV	M04	In G97 mode: start the spindle in the reverse direction at the RPM specified in the spindle RPM DRO field. In G96 mode: start the spindle at an RPM specified by the SFM DRO field and the X (Dia.) value in the X DRO field.
Stop	M05	Stop the spindle.

The FWD and REV buttons and the RPM DRO field don't operate if they're selected when:

- A G-code program is running.
- Using manual data input (MDI) commands.

There are two spindle speed command modes:

- **Constant RPM (G97)** The spindle turns at an rpm equal to the S word.

EXAMPLE

G97 S500 M03 starts the spindle at 500 rpm.

- **Constant Surface Speed (G96)** The spindle rpm varies to maintain a constant speed in terms of linear feet per minute over the workpiece. The actual spindle RPM depends on the diameter of the work being turned, increasing as diameters get smaller and decreasing as diameters get larger.

EXAMPLE

G96 S50 M03 turns the spindle on, and the linear speed at which the cutting tool's control point travels over the work piece's surface is 50 feet per minute.

Note: Because tool life and efficiency depends on surface speed (and not rpm), we recommend constant surface speed mode for all machining on a lathe.

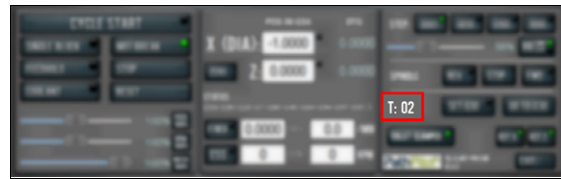


Figure 8-94: Tool DRO field.

To change the tool number (and apply its tool length offset):

1. From the PathPilot interface, on the **Offsets** tab, select the **Tool Touch** tab.



Figure 8-95: Tool DRO field.

2. In the **Tool DRO** field, type a number (the valid range is from 0-99). Then select the **Enter** key.

8.5.12 Use a G30 Position

The Go to G30 button moves the machine to a predefined G30 position. For information, see "About G30" (on the next page).

To set a G30 position:

1. Jog the machine to the desired G30 position.
2. From the **Main** tab, select **Set G30**.

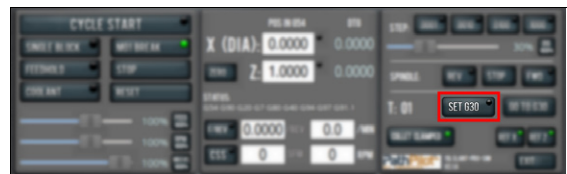


Figure 8-96: Set G30 button.

To go to a set G30 position:

- Use a G30 command in a G-code program.
- Select **Go To G30**.



Figure 8-97: 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" (on the previous page). 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 97).

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.

8.5.13 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" (below).

To manually enter commands:

1. Select the **MDI Line** DRO field.

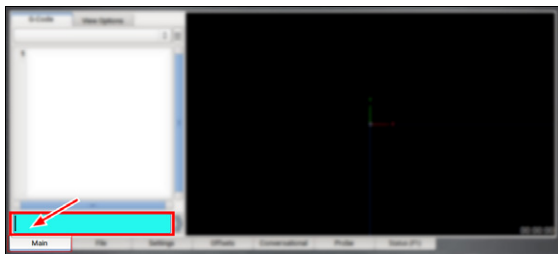


Figure 8-98: MDI Line DRO field.

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**.

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" (above).

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 MOUSE	Change the mouse preferences, like pointer speed and right- or left-hand button mapping
ADMIN NETWORK	Configure a Wi-Fi network.
ADMIN RESET_SOFT_LIMITS	Reset axis soft limits to machine defaults.
ADMIN SET_X_LIMIT	Set the X-axis soft limit.

8: PATHPILOT TOOLS AND FEATURES

8.5 Control G-Code Programs

Admin Command	Use to...
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.

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**.

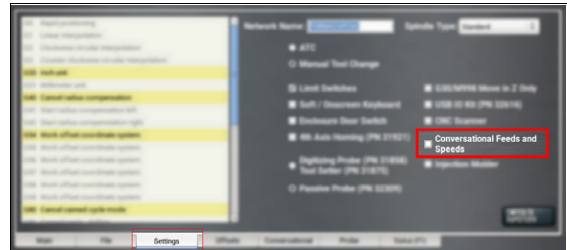


Figure 8-99: Settings tab.

2. From the **Conversational** tab, locate the **Material** dropdowns in the **Conversational DROs** group.



Figure 8-100: 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.

8.5.14 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" (on the previous page).

8.5.15 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).

6. Select **Refresh** (to the right of the **Sub-Type** dropdown).
The following machining-related DRO fields are calculated:
 - **Roughing SFM**
 - **Finishing SFM**
 - **Max Spindle RPM**
 - **Roughing (in/rev)**
 - **Finishing (in/rev)**
 - **Finishing DOC**
 - **Roughing DOC**
 - **Peck** (if drilling)

Note: After PathPilot calculates values for the machining-related DRO fields, the background turns green.

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.

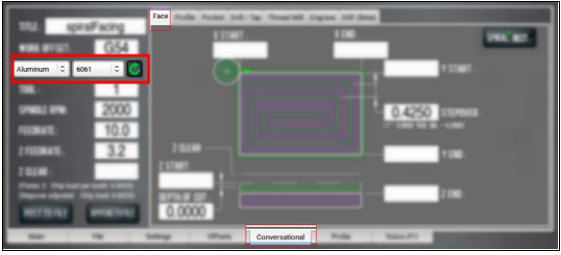


Figure 8-101: Refresh button on the Conversational tab.

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**.

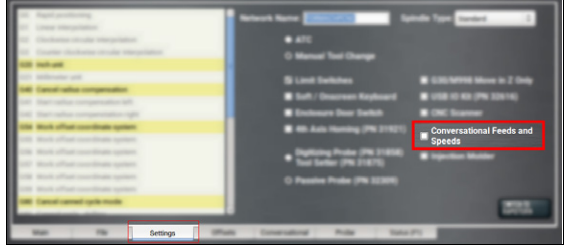


Figure 8-102: Settings tab.

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).

Manually Enter Tool Descriptions

PathPilot uses keywords and patterns in the tool description to recognize tooling features. For information, see "Tool Keywords Reference" (on the next page).

- To manually enter tool descriptions:
1. From the PathPilot interface, on the **Offsets** tab, select the **Offsets Table** tab, and identify the **Tool Table** window.
 2. Select a blank line.

8: PATHPILOT TOOLS AND FEATURES

8.5 Control G-Code Programs

3. Type a description for the tool. Descriptions are not case sensitive.

If a pattern or word in the description is recognized, PathPilot uses syntax highlighting to indicate a valid description.

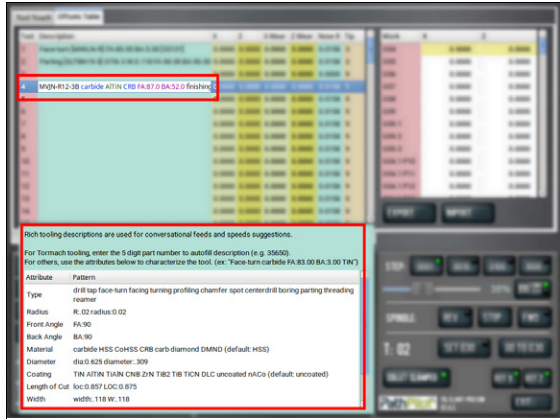


Figure 8-103: A manually-entered tool description.

Examples

To get accurate machining information, all tooling must be described with detail: the more detail, the better the results.

EXAMPLE

MVJN-R12-3B carbide AlTiN CRB FA:87.0 BA:52.0 finishing

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

- MVJN holder geometry
- Aluminum-titanium nitride coated carbide (AlTiN)
- Carbide
- 87° front angle
- 52° back angle

Using a personal description likely won't contain meaningful information for PathPilot.

EXAMPLE

Gold colored tool bit from middle drawer

This description provides very little information, and PathPilot defaults to tool bit features from the setup on the Tool Touch tab.

Automatically Generate Tool Descriptions

If you're using a Tormach tool, you can enter the part number to automatically generate tool descriptions in the Tool Table window.



Note: If you don't know the part number, you can search for the tool at tormach.com.

1. From the PathPilot interface, on the **Offsets** tab, in the **Tool Table** window, select a blank line.
2. Type the part number for the tool, like **33133**.
A full description for a right-handed profiling tool (PN 33133) displays.

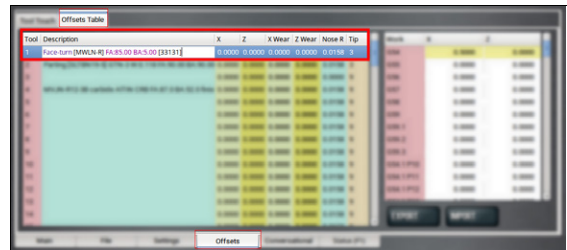


Figure 8-104: An automatically-generated tool description for a Tormach tool.

3. You must use the **Tool Touch** tab to zero the tool for the correct X offset and Y offset.

Tool Keywords Reference

PathPilot uses keywords and patterns in the tool description to recognize tooling features.

Item	Pattern	Example	Notes
type	drill, tap, face-turn, facing, turning, profiling, chamfer, spot, centerdrill, boring, parting, threading, reamer	DRILL, CHAMFER, BORING	

8: PATHPILOT TOOLS AND FEATURES

8.5 Control G-Code Programs

Item	Pattern	Example	Notes
tool radius	“R” or “radius”, followed by a colon, followed by a decimal number	R: .0158, RADIUS:0.32	No radius specified is the same as a zero radius.
front angle	“FA”, followed by a colon, followed by a decimal number	FA: 90	
back angle	“BA”, followed by a colon, followed by a decimal number	BA: 90	
tool material	carbide, HSS, CoHSS, CRB, carb, diamond, DMND	HSS, COHSS, CRB	HSS: High-speed steel; CoHSS: Cobalt high-speed steel; CRB: Carbide; No tool material specified is the same as HSS.
tool diameter	“dia” or “diameter”, followed by a colon, followed by a decimal number	DIA: .0341, DIAMETER: .750	For axial tools (drills taps, reamers, etc.).

Item	Pattern	Example	Notes
tool coating	TiN, AlTiN, TiAlN, CNB, ZrN, TiB2, TiB, TiCN, DLC, uncoated, nACo	TiN, ZRN, TiB2	No coating specified is the same as uncoated.
length of cut	“loc”, followed by a colon, followed by a decimal number	LOC:0.875	For axial tools (drills, reamers, etc.). If no length of cut is specified, a length will be assumed based on cutter diameter.
tool width	“width” or “w”, followed by a colon, followed by a decimal number	W: .118, WIDTH: .118	For grooving and parting tools.
min bore	“min-bore”, followed by a colon, followed by a decimal number	MIN-BORE: .522	
flutes	A number, followed by “FL” or “FLUTE”	4FL, 12FL, 2FLUTE	No flutes specified is the same as 2 flutes.

8.5.16 Use Cycle Counters (M30 and M99)

On the Main tab, the Tool Path display shows M30 and M99 cycle counters. They're useful to count parts completed during

8: PATHPILOT TOOLS AND FEATURES

8.5 Control G-Code Programs

unattended operation. For each M-code, there's an A and B counter. This provides more flexibility, because you can reset them to 0 independently.

For example, you could use M30 A to count parts each shift, and M30 B to count parts each week. The cycle counters persist across the controller's power cycles.

Monitor Cycle Counters

- In the **MDI Line DRO** field, type `ADMIN CYCLECOUNTER` to show or hide the counters and to reset them to 0.

Change Cycle Counter Values

The cycle counters are implemented as read-only persistent G-code numbered parameters, as detailed in the following table.

If needed, the cycle counter value can be read in G-code.

Cycle Counter	Parameter
M30 A	#5650
M30 B	#5651
M99 A	#5652
M99 B	#5653

To change a counter value explicitly, use a G10 command: `G10`

`L99 P~ Q~`

- `P~` selects the cycle counter to change. Use any of the values detailed in the following table.

Cycle Counter	P~
M30 A	0
M30 B	1
M99 A	2
M99 B	3

- `Q~` specifies the value to set the cycle counter. If `Q~` is omitted, the cycle counter is incremented by 1. For example, if you program `G10 L99 P2`, the M99 A cycle counter increments by 1.

8.6 SYSTEM FILE MANAGEMENT

To keep the files on your system backed up and organized, use the following controls:

8.6.1 Manage System Files	121
8.6.2 Create Backup Files	121
8.6.3 Restore Backup Files	122
8.6.4 Import and Export the Tool Table	123

8.6.1 Manage System Files

Use the File tab to manage system files on the PathPilot controller. For information, see "Transfer Files to and From the Controller" (page 133).

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**.

8.6.2 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.

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.

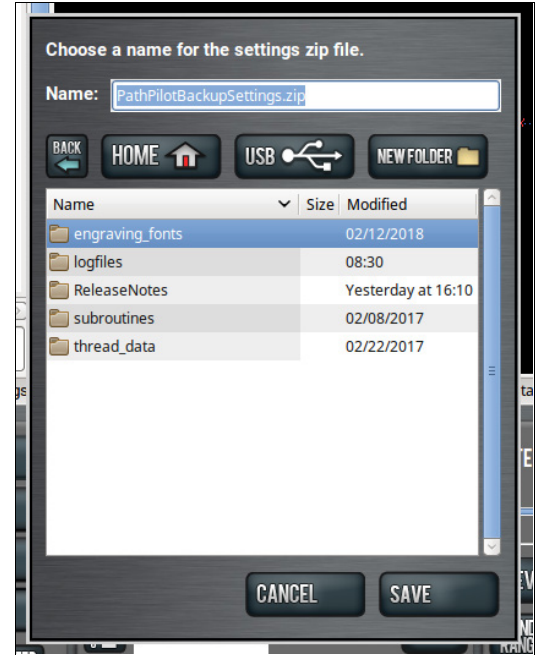


Figure 8-105: Admin Settings Backup dialog box.

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.
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.

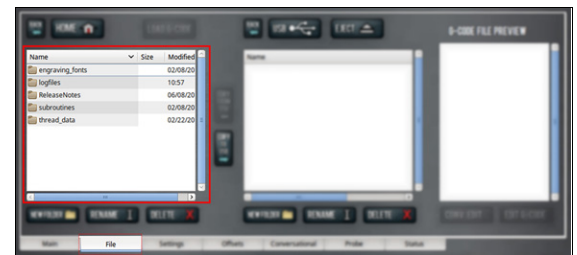


Figure 8-106: Controller Files window on the File tab.

Note: Files must have unique names. If they don't, PathPilot prompts you to overwrite or rename files, or cancel the file transfer.

8: PATHPILOT TOOLS AND FEATURES

8.6 System File Management

- To prevent errors, make sure you don't include the following folders:
 - logfiles
 - media
 - ReleaseNotes
 - subroutines
 - USB
- Select **Copy to USB**.
The files are copied and display in the **USB Files** window.
- Eject the USB drive from the PathPilot controller.
- From the PathPilot interface, select **Exit**.
- 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.
- (Optional) As an extra precaution, copy all the files onto the device.

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.

8.6.3 Restore Backup Files

- Insert the USB drive with your backup files into the PathPilot controller.
- 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.

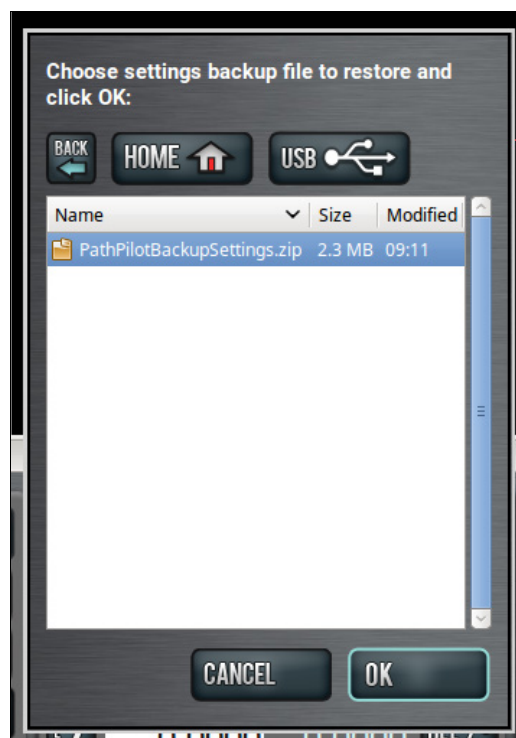


Figure 8-107: Admin Settings Restore dialog box.

- 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.

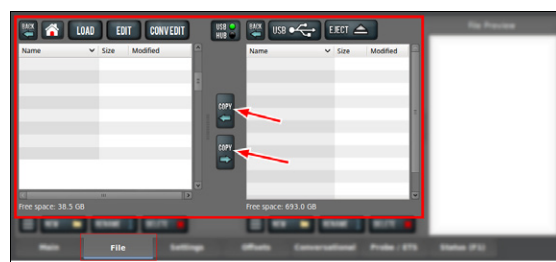


Figure 8-108: USB Files window on the File tab.



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.

6. 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.

8.6.4 Import and Export the Tool Table

You can manage the tool table using an external .csv file.

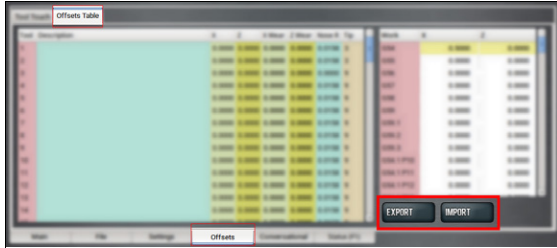


Figure 8-109: Export and Import buttons on the Offsets tab.

Import a .csv File

1. Transfer the .csv file to a USB drive.
2. Insert the USB drive into the PathPilot controller.
3. Confirm that the PathPilot controller is on.
4. From the **Offsets** tab, on the **Offsets Table** tab, select **Import**.

The **Import** dialog box displays.

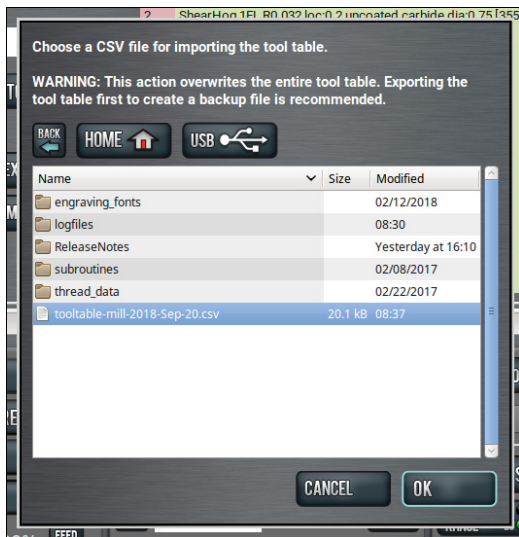


Figure 8-110: Import dialog box.

5. 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

1. From the **Offsets** tab, on the **Offsets Table** tab, select **Export**.

PathPilot generates the .csv file, and the **Export** dialog box displays.



Figure 8-111: 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

9.1 Start the Machine.....	126
9.2 Bring the Machine Out of Reset.....	127
9.3 Reference the Machine.....	128
9.4 Jog the Machine.....	129
9.5 Change the Spindle Speed Range.....	131
9.6 Load Material into the Spindle.....	132
9.7 Load G-Code.....	133
9.8 Set Work Offsets.....	134
9.9 Set Tool Geometry Offsets.....	135
9.10 Touch Off the Tool Geometry Offsets.....	136
9.11 Operate the Coolant Pump.....	138

9: BASIC OPERATIONS

9.1 Start the Machine

9.1 START THE MACHINE

Power on the machine and the PathPilot controller.

1. Turn the Main Disconnect switch to **ON** on the side of the electrical cabinet.
2. Twist out the machine's red Emergency Stop button, which enables movement to the machine axes and the spindle.
3. Press the machine's Reset button (next to the Emergency Stop button).

9.2 BRING THE MACHINE OUT OF RESET

- Select **Reset**.

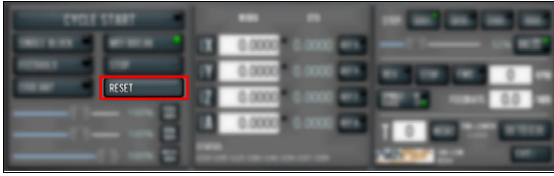


Figure 9-1: Reset button.

For more information on reset mode, see "About Reset Mode" (below).

9.2.1 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.

9: BASIC OPERATIONS

9.3 Reference the Machine

9.3 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 9-2: 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" (below).

9.3.1 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.

9.4 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.

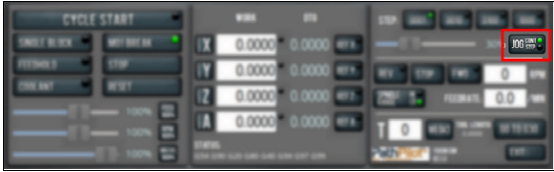


Figure 9-3: Jog button.

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.

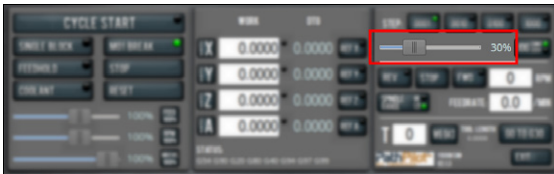


Figure 9-4: Jog Speed slider.

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.

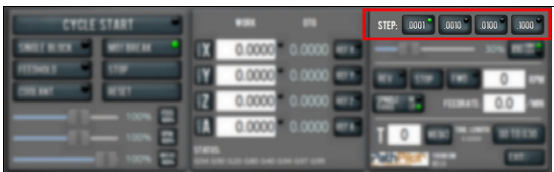


Figure 9-5: Step buttons (in G20 mode).

- 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).

9.4.1 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, Jog Shuttle, or Operator Console pendant. Whichever device you're jogging with, 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).

9.4.2 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
- The Operator Console's pendant wheel is turned in the desired direction

This is useful when you're doing things like:

- Roughly positioning the machine (for example, to move the cutting tool away from the workpiece).
- Moving the machine a certain distance at a constant speed (for example, to manually face off a workpiece).

9.4.3 About Step Jogging

While jogging in step mode, the machine moves one step at a time. 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 offsets.

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. The Operator

9: BASIC OPERATIONS

9.4 Jog the Machine

Console's pendant moves the machine in the step size that's selected with the three-position switch.

9.4.4 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 or Operator Console pendant

Axis	Direction	Keyboard Key	Jog Shuttle
X-Axis	Positive	Up Arrow	Clockwise
	Negative	Down Arrow	Counterclockwise
Z-Axis	Positive	Right Arrow	Clockwise
	Negative	Left Arrow	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 or Operator Console Pendant

Both the jog shuttle and operator console pendant provide manual jogging controls, but with slightly different physical interfaces:

• Axis Movement

- Jog Shuttle: An inner wheel provides precise, incremental jogging. Each detent (click) moves the selected axis by one jog step increment. An outer ring provides smooth, continuous jogging based on

rotation speed.

- Pendant: Provides variable-speed jogging based on rotation, similar to the shuttle's outer ring.



Note: Rotating clockwise jogs the axis in the positive direction. Rotating counterclockwise jogs the axis in the negative direction.

• Axis Selection

- Jog Shuttle: Four buttons let you toggle between the X, Y, Z, and A axes.
- Pendant: A four-position switch selects the active 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. On this machine, two of the axis selection buttons – Y-axis and A-axis – won't result in any motion if selected.

• Step Size Selection

- Jog Shuttle: A dedicated Step button toggles between available jog step sizes.
- Pendant: A three-position switch selects the jog step size.

9.5 CHANGE THE SPINDLE SPEED RANGE

This machine has two spindle speed ranges:

- **High** 350 rpm to 5000 rpm
- **Low** 180 rpm to 2500 rpm

⚠ WARNING! Entanglement / Entrapment Hazard. The machine operates under automatic control — it can start at any time and crush, cut, entangle, or pinch body parts. Always keep clear of positions on the machine where unexpected or unintended machine motion could cause harm. Before operating this machine in any way, you must verify that all operators know the location of the machine's Emergency Stop button.

To change the spindle speed range:

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.
 - d. Follow correct lockout/tagout procedures.
2. Open the spindle cabinet door.
3. Loosen the 8 mm socket head cap screw on the far side of the motor mount with a 6 mm hex wrench.
4. Lift the spindle motor up (it tilts toward you, pivoting about the remaining screw). The spindle belt slackens, which allows you to move the belt from one pulley to the other.

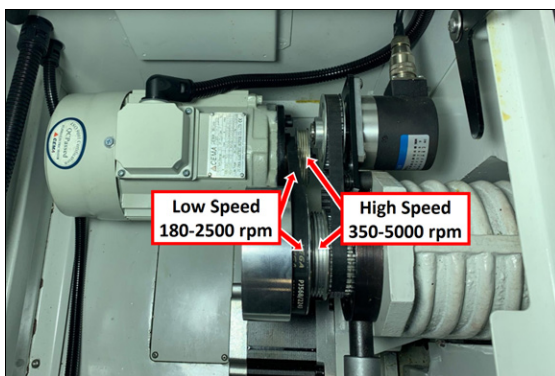


Figure 9-6: Example of the spindle belt in the low position.

5. With the spindle belt in its new position, retighten the 8 mm socket head cap screw that you loosened in Step 3.
6. Rotate the spindle by hand to visually confirm that the spindle belt is fully seated in the pulley.
7. Close the spindle cabinet door.
8. From the PathPilot interface, on the **Settings** tab, select the appropriate spindle pulley ratio option.

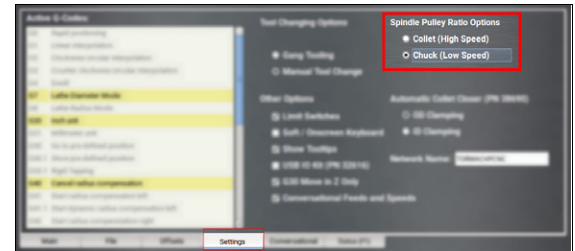


Figure 9-7: Spindle Pulley Ratio Option selections.

9: BASIC OPERATIONS

9.6 Load Material into the Spindle

9.6 LOAD MATERIAL INTO THE SPINDLE

1. Open the enclosure door and the spindle door. Verify that the latch on both doors is seated to prevent the doors from unexpectedly falling.
2. Install the draw tube through the spindle.

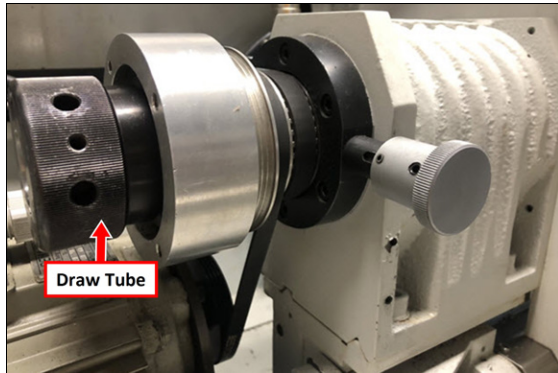


Figure 9-8: Draw tube installed into the spindle.

3. Install a 5C collet or chuck into the headstock.

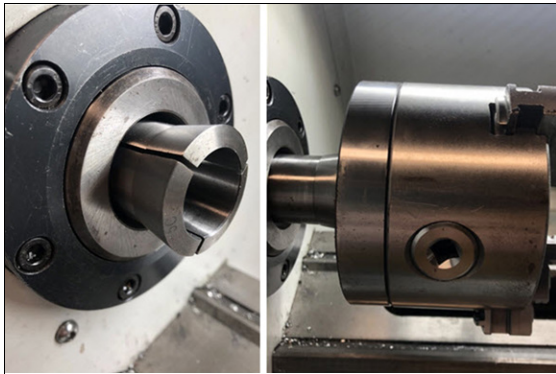


Figure 9-9: Workholding option installed.

4. Depending on which workholding option you're using, do one of the following:
 - **5C Collet** Put a piece of stock into the collet, and adjust the length of the stock in the cutting area.

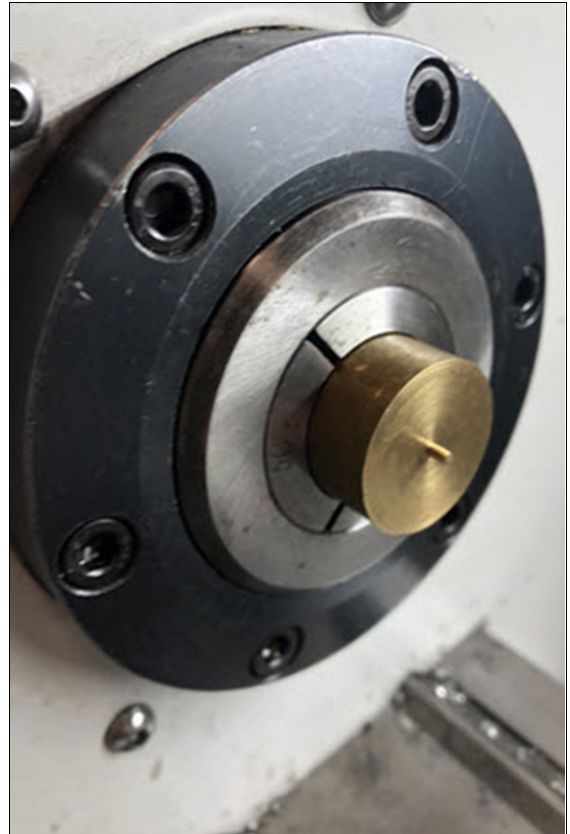


Figure 9-10: Material installed into the collet.

- **Chuck** You'll install the stock after the drawtube and the chuck are securely tightened.
5. Gently push on the spindle lock and rotate the spindle with your hand. Stop rotating the spindle once you feel the lock engage.

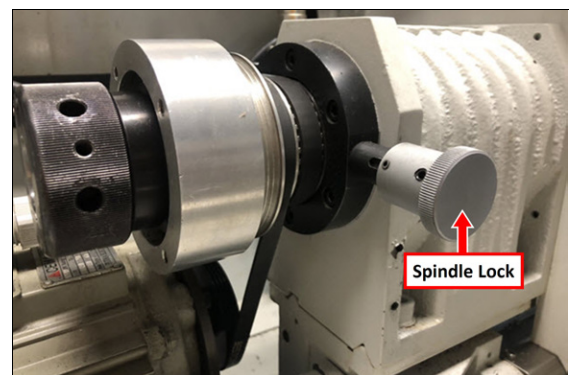


Figure 9-11: Spindle lock.

6. Hold the spindle lock with one hand and, using the other hand, tighten the draw tube with the (provided) spanner wrench until the draw tube is tight.

9.7 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**.

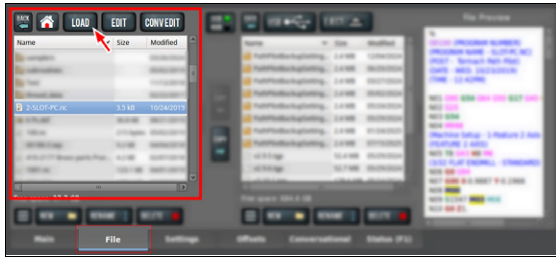


Figure 9-12: Controller Files window on the File tab.

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

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

9.7.1 Transfer Files to and From the Controller

To run a G-code program, you must transfer the files to the PathPilot controller. You can either use a USB drive or PathPilot HUB (our cloud-based simulator) to transfer files. For more information on PathPilot HUB, go to hub.pathpilot.com.

To transfer files to and from the controller:

1. Either insert a USB drive into any open USB port, or sign in to PathPilot HUB.
2. From the **File** tab, select the file to transfer (either in the **USB / HUB Files** window or the **Controller Files** window).

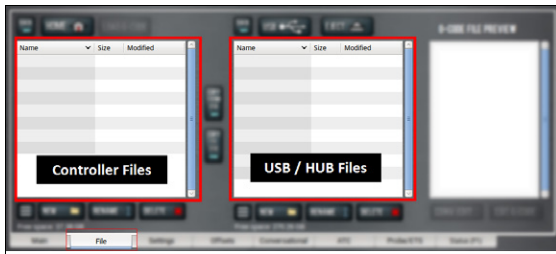


Figure 9-13: File tab.

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 ←** or **Copy →**.

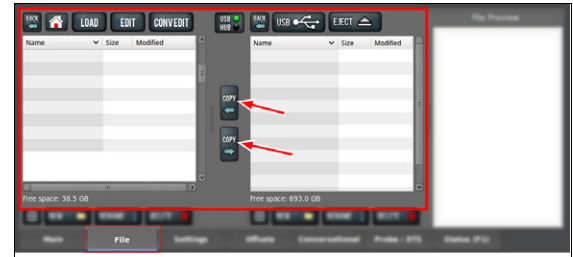


Figure 9-14: File tab.

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. If you're using a USB drive, select **Eject**. It's safe to remove the USB drive from the controller.

9: BASIC OPERATIONS

9.8 Set Work Offsets

9.8 SET WORK OFFSETS

In the the **X (Dia)** DRO field, all values are expressed in terms of the diameter of the workpiece, not the radius.

EXAMPLE
 If the active tool is 2 in. away from the spindle centerline, **4.000** displays in the **X DRO** field.

To set the current axis location to zero in the active work coordinate system:

- Select **Zero**.

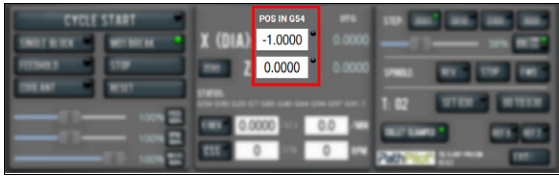


Figure 9-15: Work Offset DRO fields.

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.

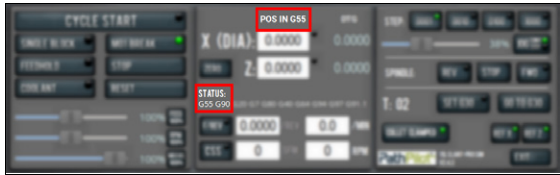


Figure 9-16: Work offset indicated in the PathPilot interface.

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" (below).

9.8.1 About Work Offsets

Work offsets allow you to think in terms of X 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
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

9.9 SET TOOL GEOMETRY OFFSETS

Before running a G-code program, PathPilot must know the geometry of the tools that are required for the program. For more information on using tool length offsets, see "About Tool Offsets" (on the next page).



Note: You can import a .csv file with tool geometry offset data. For information, see "Import and Export the Tool Table" (page 137).

To set tool geometry offsets:

1. Verify that the machine is powered on, out of reset, and the axes have been referenced.
2. From the PathPilot controller, on the **Offsets** tab, select the **Offsets Table** tab.
3. Find the **Tool Table** window.

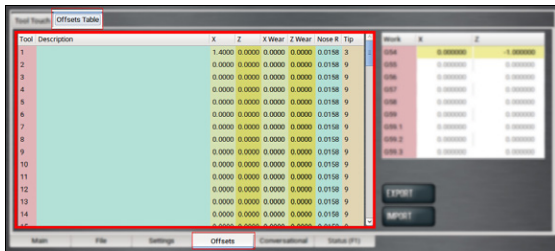


Figure 9-17: Tool Table window on the Offsets tab.

4. Touch off the tool geometry offsets. For information, see "Touch Off the Tool Geometry Offsets" (on the next page).
5. (Optional) Select a field to edit. When finished, select the **Enter** key.

9: BASIC OPERATIONS

9.10 Touch Off the Tool Geometry Offsets

9.10 TOUCH OFF THE TOOL GEOMETRY OFFSETS

On the Offsets tab, you can use the Tool Touch tab to graphically select a tool, and then touch off the tool to set the geometry offsets.



Tip! When using this method to measure your tool geometry, remember that the X zero location never changes (the spindle centerline is always $X = 0$), but the Z zero location may change depending on the length of the workpiece that is chucked into the spindle. As long as each tool is measured to a face that has been zeroed, only measure these tools one time or until you replace an insert.

To touch off the tool geometry offsets:

- From the **Offsets** tab, on the **Tool Touch** tab, select a tool.

The tools along the bottom of the screen are front tool post tools (used by machines with a quick-change tool post setup).



Note: Gang tooling setups typically use both front and rear tool post tools.

After you select the tool, PathPilot:

- Sets the tip orientation for the tool, used (along with tip radius) in cutter compensation.
- Sets the tool type (front tool post or rear tool post), used by the conversational routines to double check the user entry fields in an attempt to try to detect and prevent crashes.
- Displays the tool touch off dialog.

9.10.1 Touch X

1. Take a skim cut off of the diameter of the workpiece — just long enough to measure the cut surface with a micrometer.
2. Jog the tool away from workpiece in Z, but don't jog the machine in X.
3. Measure the diameter of the skim-cut workpiece with a micrometer.
4. In the **Touch X** DRO field, type the value that you measured in Step 3.



Note: If you're touching off a front tool post tool, verify that the value you enter is positive.

5. Select **Touch X**.
The LED comes on.

9.10.2 Touch Z

1. Jog the machine toward the part zero (usually the face of the workpiece) in the Z direction.
2. Move the tool so that its cutting edge is just touching the surface of the material and define this as $Z = 0$. Use a sheet of paper to indicate when the tool is touching the material.
3. Slowly jog the Z-axis until it's approximately 1/4 in. away from part zero on the workpiece.
4. With the paper between the tool and the workpiece, slowly jog the machine until you feel a light pull on the paper.
5. In the Touch Z DRO field, type the thickness of the piece of paper. Then select **Touch Z**.

9.10.3 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 gang tooling, different X/Z positions on the carriage).

Tool offsets are broken down into two components:

- **Geometry Offsets** Represents the distance from the work offset zero location to the tool's control point.



Note: Unlike on a mill (where G43 must be called out to apply an offset), tool geometry offsets are automatically applied with the `Txx` tool change command.

- **Wear Offsets**

The sign convention for the machine are as follows:

- Z negative is toward the spindle.
- X positive is toward the operator.

Sign convention is important when you choose the manual tool change option or the gang tooling option.

All tools mounted for use on the operator side of the workpiece are touched off using positive X (diameter) values,

and most X words in part programming for these tools have positive values.

9.10.4 Import and Export the Tool Table

You can manage the tool table using an external .csv file.

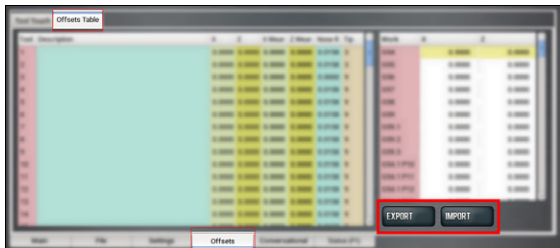


Figure 9-18: Export and Import buttons on the Offsets tab.

Import a .csv File

1. Transfer the .csv file to a USB drive.
2. Insert the USB drive into the PathPilot controller.
3. Confirm that the PathPilot controller is on.
4. From the **Offsets** tab, on the **Offsets Table** tab, select **Import**.

The **Import** dialog box displays.

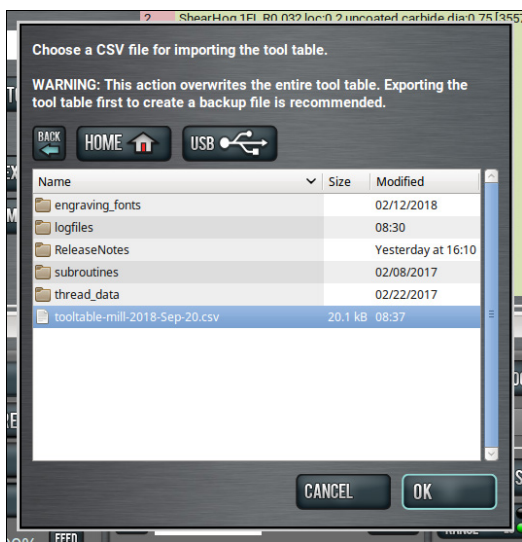


Figure 9-19: Import dialog box.

5. 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

1. From the **Offsets** tab, on the **Offsets Table** tab, select **Export**.
PathPilot generates the .csv file, and the **Export** dialog box displays.

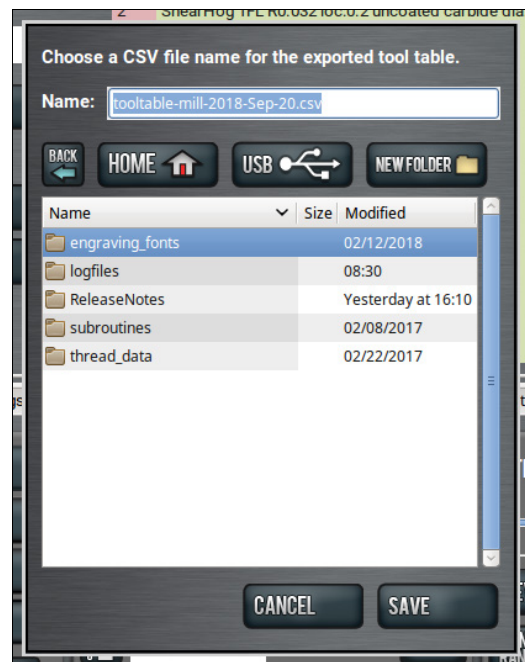


Figure 9-20: Export dialog box.

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.

9: BASIC OPERATIONS

9.11 Operate the Coolant Pump

9.11 OPERATE THE COOLANT PUMP

To turn coolant on or off:

- Select **Coolant**.



Figure 9-21: Coolant button.

For more information on turning on and off coolant, see "About Coolant" (below).

9.11.1 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.

PROGRAMMING

IN THIS SECTION, YOU'LL LEARN:

- About the languages that are understood and interpreted by PathPilot.

CONTENTS

10.1 Before You Begin	140
10.2 Programming Overview	143
10.3 Programming G-Code	148
10.4 Programming M-Code	158
10.5 Programming Input Codes	161
10.6 Advanced Programming	162

10: PROGRAMMING

10.1 Before You Begin

10.1 BEFORE YOU BEGIN

- **Referring to This Section** Use this section only for reference. To learn about the principles of the control language (so that you can write programs by hand from first principles, for example), we recommend that you consult an introductory textbook on G-code programming.
- **Creating and Editing G-Code Files** We recommend using a text editor like Gedit or Notepad++. Don't use a word processor to create or edit G-code files — it'll leave unseen codes that could cause problems or prevent a G-code file from working.

10.1.1 Definitions

Read the following sections for reference:

Linear Axes	140
Controlled Point	140
Coordinated Linear Motion	141
Feed Rate	141
Arc Motion	141
Dwell	141
Units	141
Current Position	141
Work Offsets	141
Selected Plane	141
Tool Table	141
Wear Offsets	141
Path Control Modes	142
Feed and Speed Override Controls	142
Block Delete Control	142
Optional Program Stop Control	142

Linear Axes

The X- and Z-axes are at right angles to each other. The Z-axis lies along the centerline of the spindle; distance values increase moving away from the spindle towards the right-hand side of the machine. The X-axis distance values increase as the tool moves down and toward you. The tool cutting position (controlled point) is represented by coordinates on these axes.

Controlled Point

The controlled point is the point whose position and rate of motion are controlled to make cuts. In practice, a tool is not a point – the sharper the point of a tool, the weaker it is – so a tip radius is always part of the tip profile. If this tool is in a front tool post, it can turn to a diameter moving toward the headstock (Z-) and face toward the centerline (X+). Many standard shapes of tools are available; PathPilot doesn't need to know the exact shape of the tool, but does need to know where it is used to cut and how to interpret the controlled point.

There are nine possible orientations for the cutting point, identified by numbers shown in the following image (with orientation code 3 illustrated).



Figure 10-1: Cutting point orientation graphic.

- Tools with **orientations 2 and 3** all turn to size and face. Their controlled points are the intersection of tangents to the cutting tip radius so the radius has no effect on the diameter or length of the part made. If, however, the cut is angled, then the radius means it does not cut the expected part but one a little larger than required. To machine a part corresponding to the angled tool path defined in the G-code, it is necessary to use tool-tip radius compensation (i.e., G41 or G42).
- Tools with **orientation 6** turn diameters to size and are often used for making profiles. It is difficult to accurately estimate their controlled point in the Z-direction.
- Tools with **orientations 5 and 7** turn on the face of stock and are unusual. It is difficult to accurately estimate their controlled point in the X-direction without a test cut.
- Tools with **orientation 9** are used when tool-tip compensation is applied in all moves.

Some orientations seem unusual at first glance, like a parting or grooving tool in the front tool post has orientation 1. This is because the program defines the part to the right of the tool. The size of the waste left in the chuck is unimportant.

Coordinated Linear Motion

To drive a tool along a specified path, a machining system must often coordinate the motion of both. We use the term coordinated linear motion to describe the situation in which each axis moves at constant speed and both axes move from their starting positions to their end positions at the same time. Because a lathe only has X- and Z-axes, this produces motion in a straight line (linear). In actual motions, it's often not possible to maintain constant speed because acceleration or deceleration is required at the beginning and/or end of the motion. It's feasible, however, to control the axes so that, at all times, each axis has completed the same fraction of its required motion as the other axes. This moves the tool along the same path and we also call this kind of motion coordinated linear motion.

Coordinated linear motion can be performed either at the prevailing feed rate or at rapid traverse rate. If physical limits on axis speed make the desired rate unobtainable, the axes are slowed to maintain the desired path.

Feed Rate

The rate at which the controlled point or the axes move is nominally a steady rate which may be set by you. In the interpreter, the interpretation of the feed rate is as follows unless inverse time feed rate (G93) mode is being used: the feed rate means the length in units per minute or units per spindle revolution along the programmed linear path.

Arc Motion

The axes can be controlled to move in a circular arc in the plane of the axes. While this is occurring, as in coordinated linear motion, the motions can be coordinated so that acceleration and deceleration do not affect the path.

Dwell

A machining system may be commanded to dwell (keep the axes unmoving) for a specific amount of time. The most common use of dwell is to break and clear chips or for a spindle to get up to speed. The units in which you specify dwell are seconds; a decimal value can be used to get less than one second.

Units

Units used for distances along the X- and Z-axes may be measured in millimeters or inches. Units for all other quantities involved in machine control cannot be changed. Different quantities use different specific units. Spindle speed is measured in revolutions per minute. Feed rates are expressed in current length units per minute.

Current Position

The controlled point is always at some location called the current position, and PathPilot always knows where that is. The numbers representing the current position are adjusted in the absence of any axis motion if any of several events take place:

- Length units are changed
- Tool length offsets are changed
- Work offsets are changed



Note: These events do not move the tool, instead they change its displayed position in the axis DROs.

Work Offsets

Work offsets allow you to jog the machine to an arbitrary location (like the end of a workpiece) and call that location zero. Only one work offset can be active at any given time. The default (used in this example) is G54.

Selected Plane

There is always a selected plane, which for a lathe is the XZ-plane of the machining system.

Tool Table

Zero or one tool is assigned to each slot in the tool table. The table defines the offset of the controlled point of the tool from the work offset coordinate system's origin, the wear corrections to be made to this tool offset, the tool tip radius, and the orientation of the tool tip.

The valid range of tool numbers is 0-99.

Wear Offsets

Wear offsets are values used to fine tune a part program to compensate for things like tool wear or spring back. Wear offsets are applied with the T command as well, by specifying the desired wear offset register with the last two digits of a four-digit T command. For example, T02 applies the geometry offsets for tool 2, whereas T0202 applies both geometry

10: PROGRAMMING

10.1 Before You Begin

offsets and wear offsets. There is no M6 tool change command for lathes like there is for CNC milling machines.

Path Control Modes

The lathe may be put into any one of three path control modes:

- **Exact Stop Mode** The machine stops briefly at the end of each programmed move.
- **Exact Path Mode** The machine follows the programmed path as exactly as possible, slowing or stopping if necessary at sharp corners of the path.
- **Continuous Mode with Optional Tolerance** Sharp corners of the path may be rounded slightly so that the feed rate may be kept up (but by no more than the tolerance, if specified).

Feed and Speed Override Controls

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

Block Delete Control

PathPilot allows the optional omission of blocks of code that are prefixed with the forward slash symbol (/).

Optional Program Stop Control

The optional program stop control (M01 Break) works as follows: if the button is selected, and an input line contains an M01 code, program execution is stopped at the end on the commands on that line until you select Cycle Start.

10.2 PROGRAMMING OVERVIEW

Read the following sections for a G-code overview:

10.2.1 About G-Code Programming Language	143
10.2.2 G-Code Formatting Reference	143
10.2.3 Supported G-Codes Reference	147

10.2.1 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).

10.2.2 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)

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

10: PROGRAMMING

10.2 Programming Overview

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
V	Synonymous with B
W	Synonymous with C

Letter	Description
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).
- 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 (foo) #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 (foo), 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:

1. Comment (including message)
2. Set feed rate mode (G93, G94, G95)
3. Set feed rate (F)
4. Set spindle speed (S)
5. Special I/O (M62 to M68)



Note: This is not supported.

6. Change tool (T)
7. Spindle on/off (M03, M04, M05)
8. Save State (M70, M73, restore state (M72), invalidate state (M71)
9. Coolant on/off (M07, M08, M09)
10. Enable/disable overrides (M48, M49, M50, M51, M52, M53)
11. Operator defined commands (M101 to M199)
12. Dwell (G04)
13. Set active plane (G17, G18, G19)
14. Set length units (G20, G21)
15. Cutter radius compensation on/off (G40, G41, G42)
16. Tool table offset on/off (G43, G49)
17. Fixture table select (G54 through G58 and G59 P~)
18. Set path control mode (G61, G61.1, G64)
19. Set distance mode (G90, G91)
20. Set canned cycle return level mode (G98, G99)
21. Home, change coordinate system data (G10) or set offsets (G92, G94)
22. Perform motion (G00 to G03, G12, G13, G80 to G89 as

10: PROGRAMMING

10.2 Programming Overview

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, G37/G37.1, G38.x, G73, G76, G80, G81, G82, G84, G85, G86, G88, G89}	Motion (one always in effect)
Group 2	{G17, G18, G19, G17.1, G17.2, G17.3}	Plane selection

Group	Commands	Group Description
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, M04, M05}	Spindle control
Group 8	{M07, M08, M09}	Coolant control (special case: M07 and M08 may be active at the same time)

Group	Commands	Group Description
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.

10.2.3 Supported G-Codes Reference

G-Code	Description
<u>G00</u>	Rapid linear motion
<u>G01</u>	Linear motion at feed rate
<u>G02</u>	Clockwise arc at feed rate
<u>G03</u>	Counterclockwise arc at feed rate
<u>G04</u>	Dwell
G07, G08	Diameter / radius mode <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Note: The lathe and the RapidTurn both use G07 (X positions displayed in diameter values). G08 is not used or supported in PathPilot.</p> </div>
<u>G10 L1</u>	Set tool table
<u>G10 L2</u>	Set coordinate system
<u>G10 L10</u>	Set tool table – calculated – workpiece

G-Code	Description
<u>G10 L11</u>	Set tool table – calculated – fixture
<u>G10 L20</u>	Set coordinate system
<u>G17, G18, G19</u>	Plane selection
<u>G20/G21</u>	Length units
<u>G28</u>	Return to predefined position
<u>G28.1</u>	Return to predefined position
<u>G30</u>	Return to predefined position
G33	Spindle synchronized motion (like threading)
G33.1	Rigid tapping
<u>G40</u>	Cancel cutter compensation
<u>G53</u>	Absolute coordinates
<u>G54-G59.3</u>	Select work offset coordinate system
<u>G61/G61.1</u>	Set exact path control mode
<u>G64</u>	Set blended path control mode
<u>G73</u>	High-speed peck drill
G76	Multi-pass threading cycle
<u>G84</u>	Tapping cycle
<u>G90, G90.1</u>	Arc distance mode
G91, G91.1	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92, etc.
<u>G93, G94, G95</u>	Feed rate mode
<u>G96, G97</u>	Spindle control mode
G98	Initial level return / R-point level after canned cycles

10: PROGRAMMING

10.3 Programming G-Code

10.3 PROGRAMMING G-CODE

Read the following sections as a G-code reference:

10.3.1 About the Examples Used	148
10.3.2 Rapid Linear Motion (G00).....	148
10.3.3 Linear Motion at Feed Rate (G01).....	148
10.3.4 Arc at Feed Rate (G02 and G03).....	149
10.3.5 Dwell (G04).....	149
10.3.6 Lathe Diameter Mode (G07).....	149
10.3.7 Set Offsets (G10).....	149
10.3.8 Plane Selection (G17, G18, G19).....	151
10.3.9 Length Units (G20 and G21).....	151
10.3.10 Return to Predefined Position (G28 and G28.1) ..	151
10.3.11 Return to Predefined Position (G30 and G30.1) ..	151
10.3.12 Spindle-Synchronized Motion (G33).....	152
10.3.13 Rigid Tapping (G33.1).....	152
10.3.14 Cutter Compensation (G40, G41, G42).....	153
10.3.15 Dynamic Cutter Compensation (G41.1 and G42.1).....	153
10.3.16 Absolute Coordinates (G53).....	153
10.3.17 Select Work Offset Coordinate System (G54 to G54.1 P500).....	153
10.3.18 Set Exact Path Control Mode (G61).....	154
10.3.19 Set Blended Path Control Mode (G64).....	154
10.3.20 Threading Cycle (G76).....	154
10.3.21 Tapping Cycle (G84).....	155
10.3.22 Distance Mode (G90 and G91).....	156
10.3.23 Arc Distance Mode (G90.1 and G91.1).....	156
10.3.24 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	156
10.3.25 Feed Rate Mode (G93, G94, and G95).....	156
10.3.26 Spindle Control Mode (G96 and G97).....	157

10.3.1 About the Examples Used

Many commands require axis words ($X\sim$, $Y\sim$, $Z\sim$, or $A\sim$) 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.

10.3.2 Rapid Linear Motion (G00)

For rapid linear motion, program: $G00 X\sim Y\sim$

- $X\sim$ is the X-axis coordinate
- $Y\sim$ is the Y-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

10.3.3 Linear Motion at Feed Rate (G01)

For linear motion at feed rate (for cutting or not), program:

$G01 X\sim Z\sim$

- $X\sim$ is the X-axis coordinate
- $Z\sim$ is the Z-axis coordinate

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

10.3.4 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 is normal to the XZ plane of the machine coordinate system. The direction is viewing from above the lathe.

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~ Z~ R~

For a counterclockwise arc in radius format, program: G03 X~ Z~ R~

- X~ is the X-axis coordinate
- Z~ is the Z-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.

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~ Z~ I~ K~

For a counterclockwise arc in center format, program: G03 X~ Z~ I~ K~

- X~ is the X-axis coordinate
- Z~ is the Z-axis coordinate
- I~ is the center of arc (X coordinate)
- K~ is the center of arc (Z coordinate)

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.
- The end point of the arc is the same as the current point.

10.3.5 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

10.3.6 Lathe Diameter Mode (G07)

To enter the diameter mode for axis X on a lathe, program: G07

When in the diameter mode the X axis moves on a lathe will be half the distance to the center of the lathe. For example, X1.2 turns a part with diameter 2.4.

10.3.7 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).....	150
Set Coordinate System (G10 L2).....	150

10: PROGRAMMING

10.3 Programming G-Code

Set Tool Table (G10 L10).....	150
Set Tool Table (G10 L11).....	150
Set Coordinate System (G10 L20).....	151

Set Tool Table (G10 L1)

To define an entry in the tool table, program: `G10 L1 P~ X~`

`Y~ R~ I~ J~ Q~`

- `P~` is the tool number
- `R~` is the radius of tool
- `I~` is the front angle
- `J~` is the back angle
- `Q~` is the orientation

`G10 L1` sets the tool table for the P tool number to the values of the words. A valid `G10 L1` rewrites and reloads the tool table.

EXAMPLE

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

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~`

- `P~` is the number of coordinate system to use (`G54 = 1`, `G59.3 = 9`)

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.

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~ X~ Z~ R~ I~ J~ Q~`

- `P~` is the tool number
- `R~` is the radius of tool
- `I~` is the front angle
- `J~` is the back angle
- `Q~` is the orientation

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~ Z~ R~ I~ J~ Q~`

- `P~` is the tool number
- `R~` is the radius of tool
- `I~` is the front angle
- `J~` is the back angle
- `Q~` is the orientation

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~ Z~

- P~ is the number of coordinate system to use (G54 = 1, G59.3 = 9)
- X~ is the X-axis coordinate
- Z~ is the Z-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 or Z is programmed

10.3.8 Plane Selection (G17, G18, G19)

To select the XY-plane as active, program: G17

To select the XZ-plane as active, program: G18



Note: Only operate the lathe in G18 (XZ-plane). Don't use G17 or G19.

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.

10.3.9 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.

10.3.10 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~, and Z~ words, program: G28 X~ 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 and 5163 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 store the current absolute position into parameters 5161-5163, program: G28.1

If no positions are stored with G28.1, then all axes will go to the machine origin.

Troubleshooting

It's an error if:

- Cutter Compensation is turned on

10.3.11 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~ Y~ 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: PROGRAMMING

10.3 Programming G-Code

10.3.12 Spindle-Synchronized Motion (G33)

To make a cut – like a thread – where the spindle and the tool motion are synchronized, program: `G33 X~ Z~ K~`

- `K~` is distance per revolution

EXAMPLE

Starting at `Z=0`, `G33 Z-1 K.0625` produces a 1 in. motion in Z over 16 revolutions of the spindle (for instance, to produce a 16 TPI thread).

Spindle-synchronized motion waits for the spindle index and spindle at speed signals from the machine so multiple passes line up. G33 moves end at the programmed endpoint. Thus, G33 could be used to cut tapered threads, a face scroll like in a 3-jaw chuck.



Note: K follows the drive line described by `X~ Z~`. K is not parallel to the Z axis if X endpoint is used, for example, when cutting tapered threads.

At the beginning of each G33 pass, PathPilot uses the spindle speed and the machine acceleration limits to calculate how long it takes Z to accelerate after the index pulse, and determines how many degrees the spindle rotates during that time. It then adds that angle to the index position and computes the Z position using the corrected spindle angle. That means that Z reaches the correct position just as it finishes accelerating to the proper speed, and can immediately begin cutting a good thread.

EXAMPLE

```
G90 (absolute distance mode) G0 X1 Z0.1
(rapid to position)
S100 M03 (start spindle turning)
G33 Z-2 K0.125 (move Z axis to -2 at a rate
to equal 0.125 per revolution) G0 X1.25
(rapid move tool away from work)
Z0.1 (rapid move to starting Z position) M2
(end program)
```

Troubleshooting

It's an error if:

- One axis word is not present.
- The spindle isn't turning when this command is executed.
- The requested linear motion exceeds machine velocity limits due to the spindle speed.

10.3.13 Rigid Tapping (G33.1)

For rigid tapping, program: `G33.1 Z~ K~ P~`

- `K~` is distance per revolution
- `P~` is, if present, the dwell (in seconds) before spindle reversal



CAUTION! Crash Hazard: If the X coordinate is specified non-zero or the current X coordinate is non-zero (i.e., not tapping a hole in the axis of the workpiece), a crash is likely.

A rigid tapping move consists of the following sequence.

1. A move to the specified Z coordinate, synchronized with the spindle at the given ratio and starting with a spindle index pulse.
2. When reaching the endpoint, a command to reverse the spindle (i.e., from forward to reverse).
3. Continued synchronized motion beyond the specified end coordinate until the spindle actually stops and reverses.
4. Continued synchronized motion back to the original coordinate.
5. When reaching the original coordinate, a command to reverse the spindle a second time (i.e., from reverse to forward).
6. Continued synchronized motion beyond the original coordinate until the spindle actually stops and reverses.
7. An unsynchronized move back to the original coordinate.

Spindle-synchronized motions wait for spindle index, so multiple passes line up. G33.1 moves end at the original coordinate.

EXAMPLE

```
G90 (set absolute mode)
G0 X0 Z0.100 (rapid move to starting
position)
G33.1 Z-0.750 K0.05 (rigid tap a 20 TPI
thread 0.750 deep) M2 (end program)
```

Troubleshooting

It's an error if:

- The Z-axis word is omitted.
- The spindle is not turning when this command is executed.
- The requested linear motion exceeds machine velocity limits due to the spindle speed.

10.3.14 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

10.3.15 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~ L~

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

- D~ is the tip radius multiplied by two
- L~ is the tool orientation

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
- The L word isn't in the range from 0 to 9 inclusive

10.3.16 Absolute Coordinates (G53)

For rapid linear motion to a point expressed in absolute coordinates, program: G01 G53 X~ 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

10.3.17 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

10: PROGRAMMING

10.3 Programming G-Code

- 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 134).

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).

10.3.18 Set Exact Path Control Mode (G61)

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

10.3.19 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.

10.3.20 Threading Cycle (G76)

To cut a thread in multiple passes, program: G76 P~ Z~ I~

J~ R~ K~ Q~ H~ E~ L D~

- Drive Line – A line through the initial X position parallel to the Z
- P~ is the thread pitch in distance per revolution
- Z~ is the final position of threads; at the end of the cycle the tool is at this Z position
- I~ is the thread peak offset from the drive line. Negative I values are external threads; positive I values are internal threads. Generally the material has been turned to this size before the G76 cycle.
- J~ is a positive value specifying the initial cut depth. The first threading cut is J beyond the thread peak position.
- K~ is a positive value specifying the full thread depth. The final threading cut is K beyond the thread peak position.



Note: As lathe diameter mode (G07) is always in force, the values for I, J and K are diameter measurements.

- D~ is the thread taper offset for the X-axis. This value is per side of the thread.

Signs (either negative or positive) of I, J, and K values discussed in this section assume a rear tool post configuration. If using a front tool post configuration (either quick change or gang tooling), reverse the sign of I, J, and K.

Optional Settings

- R~ is the depth degression. R1.0 selects constant depth on successive threading passes. R2.0, which is usual, selects constant area. Values between 1.0 and 2.0 select decreasing depth but increasing area. Values above 2.0 select decreasing area. Beware that unnecessarily high degression values causes a large number of passes to be used (degression = a descent by stages or steps).
- Q~ is the compound slide angle is the angle (in degrees) describing to what extent successive passes should be offset along the drive line. This is used to cause one side of the tool to remove more material than the other. A positive Q value causes the leading edge of the tool to cut more heavily. Typical values for threads with a 60° angle are 29, 29.5 or 30.

- **H~** is the number of spring passes. Spring passes are additional passes at full thread depth used to allow for any tool or workpiece deflection during the main cuts. If no additional passes are desired (for example on a work-hardening material), program H0.
- **E~** specifies the distance along the drive line used for the entry/exit taper. The angle of the taper is so the last pass tapers to the thread crest over the distance specified with E.' E0.2' will give a taper for the first/last 0.2 length units along the thread. For a 45° entry/exit taper, program E the same as K.
- **L~** specifies which ends of the thread get the taper. Program L0 for no taper (the default), L1 for entry taper, L2 for exit taper, or L3 for both entry and exit tapers. Entry tapers pause at the drive line to synchronize with the index pulse then feed in to the beginning of the taper. No entry taper and the tool will rapid to the cut depth, then synchronize and begin the cut.

The tool should be moved to the initial X and Z positions prior to issuing the G76. The X position is the drive line and the Z position is the start of the threads.

The tool pauses briefly for synchronization before each threading pass, so a relief groove is required at the entry unless the beginning of the thread is past the end of the material or if an entry taper is used.

Unless using an exit taper, the exit move (traverse to original X) is not synchronized to the spindle speed. With a slow spindle, the exit move might take only a small fraction of a revolution. If the spindle speed is increased after several passes are complete, subsequent exit moves requires a larger portion of a revolution, resulting in a very heavy cut during the exit move. This can be avoided by providing a relief groove at the exit, or by not changing the spindle speed while threading. The final position of the tool is at the end of the drive line. A safe Z move is needed with an internal thread to remove the tool from the hole.

Troubleshooting

It's an error if:

- Other axis words, such as X-, are specified
- The R- degression value is less than 1.0
- All the required words are not specified
- P~, J~, K~ or H~ is negative
- E~ is greater than half the drive line length

10.3.21 Tapping Cycle (G84)

The G84 cycle is intended for tapping. This cycle rotates the spindle clockwise to tap a pre-drilled hole; when the bottom of the hole is reached, the spindle rotates in the reverse direction and exits the hole.

Program: G84 X~ Y~ Z~ R~ P~ F~

- X~ is the position of the hole on the X-axis
- Y~ is the position of the hole on the Y-axis
- Z~ is the depth, tapping from the R-plane to the Z-depth
- R~ is the position of the retract plane (R-plane)
- P~ is the number of seconds to dwell
- F~ is the feed rate

The G84 cycle is as follows:

- x **Step 1:** Preliminary canned cycle motion.
- x **Step 2:** Start the spindle forward.
- x **Step 3:** Move the Z-axis at the programmed feed rate (F~) to the Z-depth.
- x **Step 4:** Reverse the spindle.
- x **Step 5:** Dwell for the P number of seconds.
- x **Step 6:** Retract the Z-axis at the programmed feed rate (F~) to the R-plane.

This cycle uses a P word, where P specifies the number of seconds to dwell. The P word is optional – if it is not included, PathPilot calculates a dwell for you (half of a second per 1000 RPM).

Spindle speed must be commanded before calling a G84 cycle. Feed rate override is ignored during a tapping cycle. Feedhold is ignored until the return operation is executed. After the tapping operation is completed, either a G98 or G99 command controls the return height – G99 returns the tool to the R-plane; G98 returns the tool to the initial height.

10: PROGRAMMING

10.3 Programming G-Code

EXAMPLE

```
N40 T51 G43 H51 M6
N45 S400 M3
N50 G54
N55 M8
N65 G0 X0.5 Y-0.75
N70 G43 Z0.6 H51
N80 G0 Z0.2
N85 S400
N90 G98 G84 X0.5 Y-0.75 Z-0.605 R0.2 F20.
N95 X1.0 Y -1.25
N100 G80
N105 G0 Z0.6
```

- $X\sim$ 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

G92 reassigns the current controlled point to the coordinates specified by the axis words ($X\sim$, $Y\sim$, $Z\sim$, and/or $A\sim$). 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).

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

10.3.25 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

10.3.22 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.

10.3.23 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.

10.3.24 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 $X\sim$ $Y\sim$ $Z\sim$ $A\sim$

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

10.3.26 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

10.4 Programming M-Code

10.4 PROGRAMMING M-CODE

Read the following sections for reference:

10.4.1 Supported M-Codes Reference.....	158
10.4.2 Program Stop and Program End (M00, M01, M02, and M30).....	158
10.4.3 Display Information and Capture Images During an M00 or M01 Break.....	159
10.4.4 Spindle Control (M03, M04, and M05).....	159
10.4.5 Coolant Control (and M09).....	159
10.4.6 Automatic Collet Closer Control (M10 and M11).....	159
10.4.7 Override Control (M48 and M49).....	160
10.4.8 Feed Override Control (M50).....	160
10.4.9 Spindle Speed Override Control (M51).....	160
10.4.10 Set Current Tool Number (M61).....	160
10.4.11 Call Subroutine (M98).....	160
10.4.12 Return from Subroutine (M99).....	160
10.4.13 USB Camera Control (M301, M302, M303).....	160

10.4.1 Supported M-Codes Reference

M-Code	Description
M00	Program stop
M01	Optional program stop
M02	Program end
M03, M04	Rotate spindle clockwise/counterclockwise
M05	Stop spindle rotation
M07, M08	Coolant on
M09	All coolant off
M10, M11	Unclamp/clamp automatic collet closer
M30	Program end and rewind
M48	Enable speed and feed override
M49	Disable speed and feed override
M64	Activate output relays
M65	Deactivate output relays
M98	Call subroutine

M-Code	Description
M99	Return from subroutine/repeat
M301, M302, M303	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.

10.4.3 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 98).

10.4.4 Spindle Control (M03, M04, and M05)

To start the spindle turning clockwise at the currently programmed speed, program: `M03`

To start the spindle turning counterclockwise at the currently programmed speed, program: `M04`

The speed is programmed by the `S` word.

To stop the spindle from turning, program: `M05`

It's okay to use `M03` or `M04` 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` or `M04` when the spindle is already turning, or to use `M05` when the spindle is already stopped. If a collet closer is installed, the spindle cannot be started while the collet closer is open.

10.4.5 Coolant Control (and M09)

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 Automatic Collet Closer Control (M10 and M11)

To unclamp the collet closer, program: `M10`



Note: The collet closer cannot be opened while the spindle is running.

To clamp the collet closer, program: `M11`

10: PROGRAMMING

10.4 Programming M-Code

10.4.7 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.

10.4.8 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).

10.4.9 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.

10.4.10 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

10.4.11 Call Subroutine (M98)

To call a subroutine, program: `M98 P~ L~` or `M98 ~P ~Q.`

- `L~` or `Q~` gives the number of times that the subroutine is to be called before continuing with the line following the M98. If this word is omitted, value defaults to 1.

The program must contain a letter O line with the number of the P word of the call (for instance O1, O125, O777). This O line is a label which indicates the start of the subroutine. The O line, and the associated G-code, is normally placed at the end of the program with other subroutines following an M2, M30 or M99, so it is not reached directly by the flow of the program.

By using parameters, values, or incremental moves, a repeated subroutine can make several roughing cuts around a complex path or cut several identical objects from one piece of material.

Subroutine calls may be nested: a subroutine may contain a M98 call to another subroutine. As no conditional branching is permitted, it is not meaningful for subroutines to call themselves recursively.

10.4.12 Return from Subroutine (M99)

To return from a subroutine, program: `M99`

Execution continues after the M98 G-code which called the subroutine.

10.4.13 USB Camera Control (M301, M302, M303)

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.

To begin a video recording by an attached camera,

program: `M301`

To stop a video recording by an attached camera, program: `M302`

To take a picture with an attached camera, program: `M303`
For more information, see "Use a USB Camera" (page 98).

10.5 PROGRAMMING INPUT CODES

Read the following sections for reference:

10.5.1 Feed Rate (F)	161
10.5.2 Spindle Speed (S)	161
10.5.3 Change Tool Number (T)	161

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 99 is used

To select a tool, program: **T~~**

The first number indicates the tool to be selected and the geometry offsets entry for it in the tool table. The second number, which is optional, indicates the line of the tool table giving the wear offsets to be applied.



EXAMPLES

- **T03**: Selects tool number 3 with geometry offsets from line 3 of the tool table.
- **T0303**: Selects tool number 3, and applies geometry and wear offset values from line 3 of the tool table.
- **T0309**: Selects tool number 3 and wear offsets from line 9 of the tool table (this isn't often used in practice).

If you select Manual Tool Change, the machine stops and displays the required tool number while Cycle Start LED flashes. The program continues when you select clicking Cycle Start.

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	162
10.6.2 Expressions	164
10.6.3 Subroutines	165

10.6.1 Parameters

Read the following sections for reference:

Parameters Reference	162
Numbered Parameters Reference	163
Subroutine Parameters Reference	163
Named Parameters Reference	163

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
- 5413: Current tool orientation
- 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.

10: PROGRAMMING

10.6 Advanced Programming

You can't access a local parameter outside of its subroutine. This is so two subroutines can use the same parameter names without fear of one subroutine overwriting the values in another.

`#<_global_named_parameter_here>` (i.e., name starting with an underscore) is a global named parameter. They are accessible from within called subroutines and may set values within subroutines that are accessible to the caller. As far as scope is concerned, they act just like regular numeric parameters. They are not made persistent by storage in a file. 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 Reference	164
Functions Reference	164

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	165
Conditional Subroutines Reference	166
Repeating Subroutines Reference	166
Looping Subroutines Reference	166

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 their 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
```

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 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
- 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	170
11.2 Maintenance Schedules	171
11.3 Clean the Coolant System	172
11.4 Examine the Enclosure Windows	174
11.5 Examine the Spindle Belt	175
11.6 Lubricate the Machine	176
11.7 Prevent Rust	177

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.

11.1.3 Coolant Maintenance Procedures

- ✓ Wear rubber 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 205). 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 ways and carriage of chips and other swarf with a chip brush.
- Clean the chip basket in the coolant tank.
- Examine the coolant level, concentration, and condition, and refill as necessary.
- 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.

- Use a rust inhibitor on all exposed, non-lubricated, non-painted metal surfaces.



Note: Don't use rust inhibitor on the ball screws or the slideways.

- If you have any accessories that require compressed air, verify that the FRL Filter-Regulator-Lubricator has at least 90 psi compressed air, and that the lubricator has air tool oil above the bottom of the plastic tube.

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 and, if necessary, adjust the draw tube tension when using the 5C automatic collet closer.

11.2.3 Monthly

- Clean the electrical cabinet vents of dust with a clean cloth or compressed air.
- Clean the fine swarf from the coolant tank.
- Clean behind the bed casting to remove chips and other swarf with a chip brush.
- Dispose of oil in the drain bucket.

11.2.4 Semi-Annually

- Examine the way covers for wear.
- Examine the oil system for blockages.
- Examine lubrication hoses for wear.
- Examine coolant hoses for wear and leaks.
- Examine the Z-axis way wipers on the carriage for wear.

11.2.5 Annually

- Verify that the limit switches are properly functioning.
- Check the machine's backlash.
- Examine the machine for any surfaces that have had paint removed from coolant or chips. Use touch-up paint if necessary.

11: MACHINE MAINTENANCE

11.3 Clean the Coolant System

11.3 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:
 - 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 the machine with a desktop coolant system, remove the rectangular drain cover on the base of the machine, and inspect the inside of the drip tray for accumulation of chips or other debris. This helps prevent clogs of the drain line, which can result in coolant backing up into the enclosure.

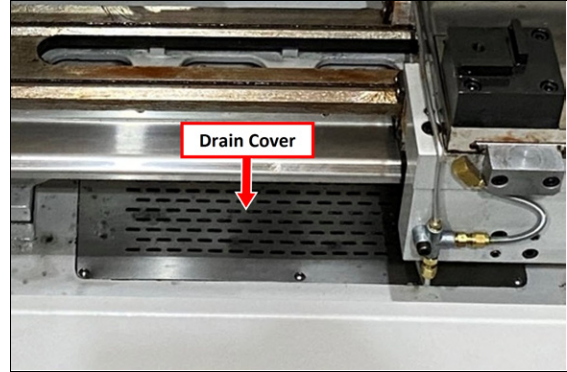


Figure 11-1: Drain cover on the base of the machine.



Note: If the coolant flow is too high, it's possible it could overwhelm the drain. Use the valve on the side of the magnetic coolant base to adjust the coolant as necessary.

- 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.

11.3.1 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.

11.3.2 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: MACHINE MAINTENANCE

11.4 Examine the Enclosure Windows

11.4 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: Window, 8L, Enclosure Door (PN 50513)

11.5 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 Spindle Belt, 8L (PN 37832).

11: MACHINE MAINTENANCE

11.6 Lubricate the Machine

11.6 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).

- Eight sliding surfaces
 - Four on the X-axis
 - Four on the Z-axis
- Two ball screw nuts

11.6.1 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.

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.

11.6.2 Lubrication System Reference

The machine's lubrication system distributes oil to the following 10 points:

11.7 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 all exterior surfaces with a mild cleaner.
- Only use flood coolant that is recommended for machining (see "Cutting Fluid Reference" (page 172)).
- 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).

11.7.1 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.



TROUBLESHOOTING

IN THIS SECTION, YOU'LL LEARN:

- About common causes of failure in this machine, and our recommendations for diagnosing and correcting them.



WARNING! Electrocution Hazard - Electrical Cabinet: Do not make or disconnect connections under power.



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

CONTENTS

12.1 Troubleshooting Safety	180
12.2 Getting Help	181
12.3 Required Tools	182
12.4 Frequently Found Problems	183
12.5 Power Distribution Subsystem	184
12.6 Control Power Subsystem	186
12.7 Axes Drive Subsystem	188
12.8 Spindle Drive Subsystem	196
12.9 Operator Console Troubleshooting	201

12: TROUBLESHOOTING

12.1 Troubleshooting Safety

12.1 TROUBLESHOOTING SAFETY

Read and understand the following safety messages before beginning any troubleshooting procedures.

- ✔ Take things slow and be extra cautious. During troubleshooting, you're exposed to more hazards than during normal operation. For example, you may have to do an electrical test on a live circuit, remove guards, or override a safety switch to make an observation.
- ✔ Power off the machine and disconnect the pneumatic supply before doing any troubleshooting procedures.
- ✔ When appropriate, lockout/tagout the Main Disconnect switch and the pneumatic supply line before doing any troubleshooting procedures.

12.2 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 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" (below).
 - 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.

12: TROUBLESHOOTING

12.3 Required Tools

12.3 REQUIRED TOOLS

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

- #2 Phillips screwdriver
- #3 Phillips screwdriver
- 1/8 in. flat-blade screwdriver
- 3/16 in. flat-blade screwdriver
- Digital multimeter that can test for:
 - Vac volts (up to 300 Vac)
 - Vdc volts (up to 100 Vdc)
 - Resistance (from 0 to 1M ohms)
 - Hz (frequency)
- Electrical safety gloves
- Measuring tools (like a tape measure, calipers, or dial indicator with magnetic base)
- Metric hex wrench set
- Needle nose pliers
- Trouble light, headlamp, or flashlight
- Wire stripper

12.4 FREQUENTLY FOUND PROBLEMS

There are several frequently found problems with all electromechanical machinery. Among the problems that have occurred, we've found that the following are more frequent than others:

- **Loose Wires**



Note: Before you begin the two-finger pull test, we recommend taking photos of the inside of the electrical cabinet to serve as a visual reference while reconnecting any loose wires.

To determine if a wire is loose, use the two-finger pull test:

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.
 - d. Follow correct lockout/tagout procedures.
2. With your thumb and index finger, hold the wire close to its termination point, and gently tug each wire.
3. If the wire comes loose, re-terminate and reconnect it before moving on to other wires.

- **Poor Cable Connections**

An improperly seated cable may allow some functions to work but cause others not to. We have found that the ribbon cables' plug connections can become loose during the shipping process.

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.
 - d. Follow correct lockout/tagout procedures.
2. Unplug and firmly reseal connectors.

12: TROUBLESHOOTING

12.5 Power Distribution Subsystem

12.5 POWER DISTRIBUTION SUBSYSTEM

Electrical power is run through a single power cord to the Main Disconnect switch. This switch controls all power to the machine, the coolant pump, and the PathPilot controller.

To troubleshoot the power distribution subsystem, read the following:

12.5.1 The Controller Won't Power On	184
12.5.2 The Coolant Pump Won't Run	184

12.5.1 The Controller Won't Power On

Cause: The PathPilot controller isn't plugged in to an outlet.

Probability	How-To Steps	Need More?
High	➤ Reseat the power cord connection at both the outlet and the controller.	It's possible that the power cord could become loose from movement.

Cause: The CB3 circuit breaker tripped.

Probability	How-To Steps	Need More?
Medium	➤ If the monitor and the controller both don't have power, examine the power cords for damage or exposed wires, and then reset the CB3 circuit breaker.	CB3 affects the monitor and the controller.

Cause: The video cable is disconnected from the PathPilot controller.

Probability	How-To Steps	Need More?
Low	➤ Reseat the video cable connection at both the monitor and the PathPilot controller.	If the monitor is disconnected, it could seem like a controller power issue.

12.5.2 The Coolant Pump Won't Run


Cause: Fuse F1 on the machine control board is blown.

Probability	How-To Steps	Need More?
Medium	<ol style="list-style-type: none"> 1. Unplug the coolant pump power cord and examine it for damage. 2. Use a digital multimeter to verify the motor's winding-to-winding and winding-to-ground resistances. If any winding-to-winding resistance measurement is a short (near zero resistance) or an open circuit or if the winding-to-ground resistance is a short, the coolant pump must be replaced. 3. Replace the fuse with a 3A slow-blow fuse, power on and reset the machine, and select the Coolant button in PathPilot. Use a digital multimeter to test the power at the XS2 receptacle. If the fuse blows again with the pump plugged in, replace the pump. 	If the coolant pump shorts, the 3 A, slow-blow, F1 flood coolant fuse on the ECM1 control board blows before CB3 trips.

Cause: The coolant pump isn't plugged in to an outlet.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Examine the connection at the outlet. 	It's possible that the power cord could become loose from movement.

Cause: The coolant control relay on the machine control board failed.

Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> 1. From the PathPilot interface, toggle the Coolant button to On. 2. Examine the coolant receptacle for power. If there's no power, and fuse F1 has continuity, it's likely that the K3 relay failed. 3. Power off the machine (see "Power off the Machine" (page 52)). Then, install a short jumper between wires J5-1 and J5-2. 4. Power on the machine (see "Power on the Machine" (page 49)). If the pump runs when the machine is powered on, replace the machine control board. 	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). Then, install a short jumper between wires 109 and 125 on the machine control board. 2. Power on the machine (see "Power on the Machine" (page 49)). If the pump runs when machine is powered on, it's likely that the problem is the K3 relay. <div style="border: 1px solid black; padding: 10px; margin-top: 10px;"> <p>Note: This relay will fail if too high of a current draw passes through it. This is common  when electrical loads much larger than the coolant pump are plugged into the coolant outlet.</p> </div>

12: TROUBLESHOOTING

12.6 Control Power Subsystem

12.6 CONTROL POWER SUBSYSTEM

To troubleshoot the control power subsystem, read the following:

12.6.1 The Machine Won't Power On 186

12.6.1 The Machine Won't Power On

Cause: The Emergency Stop button is pushed in.		
Probability	How-To Steps	Need More?
High	<ul style="list-style-type: none"> ➤ Twist out the Emergency Stop button and press the Reset button. 	The Reset button doesn't illuminate until after you: <ol style="list-style-type: none"> 1. Twist out the Emergency Stop button. 2. Press the Reset button.
Cause: The DC power supply PS1 is defective.		
Probability	How-To Steps	Need More?
Medium	<ul style="list-style-type: none"> ➤ Measure for 24 Vdc nominal between wires 401 and 400. 	Before you replace the DC power supply, power off the machine (see "Power off the Machine" (page 52)).
Cause: The Main Disconnect switch is in the Off position.		
Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Examine the Main Disconnect switch. If it's not already in the On position, turn it on. 	If needed, measure for 115 Vac nominal between wires 101 and 100/N.
Cause: The mains breaker is turned off.		
Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Examine the breaker. If it's not already on, turn it on. 	
Cause: The CB1 and/or CB2 circuit breaker tripped.		
Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> 1. Measure for 115 Vac nominal between wires 105 and 100/N. 2. Measure for 115 Vac nominal between wires 107 and 100/N. 	Before you reset the tripped breaker, power off the machine (see "Power off the Machine" (page 52)).

Cause: The contactor K1 is defective.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> • If the contactor's red LED is on, the contactor is latched. Use a digital multimeter to examine the power at K1-1 and wire 115. • If the contactor's LED is not on, press and hold the Reset button and observe the LED on K1: <ul style="list-style-type: none"> ◦ If it's on, K1 has a latching circuit issue. ◦ If it's not on, K1 may have a coil issue. 	<p>Contactor K1 energizes the DC-BUS board, which provides 65 Vdc to the machine. K1 can fail by not energizing the coil, or the contacts could fail.</p>

12: TROUBLESHOOTING

12.7 Axes Drive Subsystem

12.7 AXES DRIVE SUBSYSTEM

The axis motors are used to move the X- and Z-axis. The motors are powered by electronic driver modules (also referred to as axis drivers) which receive control signals from the control board. The electronic driver modules are powered by the DC-BUS board. Travel limits are established by limit switches when the machine is referenced.

To troubleshoot the axes drive subsystem, read the following:

12.7.1 All Axes Won't Move When Commanded	188
12.7.2 One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move	189
12.7.3 Axis Movement is Noisy	191
12.7.4 Can't Reference All Axes	192
12.7.5 Lost Motion on Axis Travel	193

12.7.1 All Axes Won't Move When Commanded

Cause: Control signals aren't reaching the electronic driver modules.

Probability	How-To Steps	Need More?
High	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 2. Examine the connectors at the J6 connection at the machine control board, and the ribbon cables at the axes drivers: <ol style="list-style-type: none"> a. Remove the connectors, and inspect them for any bent pins or discoloration. b. Firmly reseal the connectors. 	Examine the ribbon cable 423 from the control board (J6) to the axes motors.

Cause: The DC-BUS board is malfunctioning.

Probability	How-To Steps	Need More?
Medium	<p>The loss of DC-BUS board power to one or more axes is likely if the axes driver LEDs are not on, if they're dim, or if they're a color other than green.</p> <ul style="list-style-type: none"> ➤ Examine the axis status LEDs on the axis drivers. 	For information, see "There's a blown fuse on the DC-BUS board" in "One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move" (on the next page).

Cause: PathPilot isn't commanding the move, or there's a controller problem.

Probability	How-To Steps
Low	<ol style="list-style-type: none"> 1. Jog the axes and, from the PathPilot interface, examine the value displayed in their DRO fields. If the position doesn't change while you're jogging, there's a problem with the controller. 2. Select the Main tab, then, on the keyboard, select the Esc key. 3. Try to jog the axes again, and examine the value displayed in their DRO fields. 4. If the problem persists, restart the controller.

12.7.2 One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move

Cause: There are loose wires or ribbon cables.

Probability	How-To Steps
High	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 2. Examine the connection of the J6 ribbon cable and the power wires from the DC-BUS board to the affected driver. 3. Power on the machine (see "Power on the Machine" (page 49)) and test for operation.

Cause: There's a defective or malfunctioning axis driver.

Probability	How-To Steps	Need More?
Medium	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 4. On the malfunctioning axis driver, replace the ribbon cable connector for the control signals and the motor/DC supply connector with those from a functioning axis driver. 6. Power on the machine (see "Power on the Machine" (page 49)). 7. Jog the malfunctioning axis in both directions. If the malfunctioning axis now moves properly, then it's likely that the malfunctioning axis driver is defective. 8. Jog the functioning axis in both directions. A defective malfunctioning axis driver is confirmed if the previously functioning axis has the same problem. 	<p>Swapping control signals between axis drivers is very helpful during troubleshooting (there are at least three identical axis drivers in this subsystem).</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Note: If control signals are switched from the driver on the non-functioning axis to a driver on a functioning axis, the end of travel limit switch on the non-functioning axis won't work. Take care to avoid reaching the end of travel when moving an axis.</p> </div>

Cause: There's a blown fuse on the DC-BUS board.

Probability	How-To Steps	Need More?
Medium	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 2. Remove the cover from the DC-BUS board. 3. Measure the continuity on each fuse with a multimeter. Then, visually inspect each fuse. If a fuse is blown, replace it with an equivalent fuse. 	<p>A blown fuse usually is the result of a defective drive or wiring. Inspect the axis' wiring carefully and repair any damage observed. If you replace a fuse and it immediately blows, it's likely a defective axis drive or its wiring.</p>

Cause: There's a loose axis motor coupling.

Probability	How-To Steps
Low	<ul style="list-style-type: none"> • Jog the axis and listen to determine if you can hear the motor run. • Remove the cover plate over the coupling and make witness marks to determine if the motor's turning but the screw isn't.

12: TROUBLESHOOTING

12.7 Axes Drive Subsystem

Cause: There's a defective motor or motor connection.

Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 4. Remove the motor leads from the axis driver, and examine the motor for signs of coolant contamination. 5. Measure the resistance of windings at the green connector on the axis driver (see Motor Resistance Reference). 6. If the resistance is out of range, carefully check the wiring: <ol style="list-style-type: none"> a. Locate the axis motor connector near the motor and repeat the resistance test. b. If the resistance is out of spec again, then the motor is defective. c. If the resistance is within spec, then inspect the cable and connectors between the axis motor and axis driver. 	When making resistance measurements on motors and other devices with low resistance, always take a tare reading on the meter before doing the resistance measurement on the motor or device.

Cause: The gibs are too tight or too loose.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Adjust the gibs. 	Gibs that are too tight result in too much friction in the ways. Gibs that are too loose can cause binding.

Cause: Oil isn't getting to the ways and ball screw.

Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> 1. Examine the oil level in the oiler. 2. Investigate the oiling system for lack of oil and/or plugged lines (See Tormach service bulletin SB0031). 	

Cause: There's oil residue from long-term storage.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Repeatedly pump oil and slowly jog the axis. 	

Cause: There's debris on the ball screw.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Clean the ball screw. 	Debris on the ball screws can rapidly accelerate wear and reduce the lifespan of your machine. Evaluate if additional measures are required to protect your machine's ball screws and ways (like supplemental dust collection).

Cause: There's a thermal trip or an electrical short on an axis driver.

Probability	How-To Steps
Low	<ol style="list-style-type: none"> 1. Examine the LEDs on the axis drivers. If there's a red LED on the driver, that means it's tripped. 2. Power the machine on and off, and the trip should reset. 3. If the problem continues, examine the wiring for shorts and test the motor resistance. (For more information, see "There's a defective motor or motor connection." earlier in this section.) 4. If the problem continues, replace the axis driver.

12.7.3 Axis Movement is Noisy

Before You Begin

⚠ WARNING! Electrocutation Hazard: When servicing the machine from inside the electrical cabinet, always use caution. Points in the electrical cabinet have high voltages that can electrocute or shock you. Even after you've powered off the machine, electronic devices in the electrical cabinet may retain dangerous electrical voltages. Only qualified electrical machinery technicians should perform maintenance or troubleshooting procedures inside the electrical cabinet while power is still on.

Some procedures in this section require servicing the machine from inside the electrical cabinet. Before you begin, you must identify a qualified electrical machinery technician to perform the procedures.

Cause: There's a loose wire connection or failed connector.

Probability	How-To Steps	Need More?
High / Low	➤ Power off the machine (see "Power off the Machine" (page 52)). Then, tighten all screw connections.	Examine the green power connector for signs of overheating.

Cause: There's a defective axis driver module.

Probability	How-To Steps	Need More?
Medium	➤ See "One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move" (page 189).	There have been cases of a noisy axis relating to a defective axis driver. This may be temperature-dependent.

Cause: There's loose sheet metal.

Probability	How-To Steps	Need More?
High	➤ Feel for vibrating sheet metal.	Loose sheet metal is mistakenly diagnosed as a noisy axis motor. On some systems, certain axis motor speeds can cause audible vibration.

12: TROUBLESHOOTING

12.7 Axes Drive Subsystem

Cause: The C1 (DC-BUS) capacitor is defective.

Probability	How-To Steps
Low	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). Then, unplug the green power connectors on all of the axis drivers (X, Y, Z, and A). 2. With the electrical cabinet door open, power on the machine (see "Power on the Machine" (page 49)). 3. Examine the green LED on the DC-BUS board, and then twist out the Emergency Stop button and press the Reset button. The green LED should come on. 4. Push in the Emergency Stop button. If the LED goes out in two seconds or less, the capacitor is defective and must be replaced. If the LED takes five seconds or more to go out, the capacitor is OK. 5. If the results are not conclusive, power off the machine (see "Power off the Machine" (page 52)). Then, unplug the green power connectors from the axis drivers (if they're not already unplugged). 6. Power on the machine (see "Power on the Machine" (page 49)). Then, carefully measure DC voltage on wires 202 (common) and 203 on the DC-BUS board. If there's a DC voltage of a nominal 65 Vdc (55-75), this indicates the capacitor is OK. If there's a DC voltage of a nominal 40 Vdc (35-45), this indicates the capacitor is defective.

12.7.4 Can't Reference All Axes

Cause: The machine must be reset.

Probability	How-To Steps	Need More?
High	<ul style="list-style-type: none"> ➤ From the PathPilot interface, select Reset. 	For information, see "Bring the Machine Out of Reset" (page 127) and "About Reset Mode" (page 127).

Cause: The machine is stuck on a limit switch.

Probability	How-To Steps	Need More?
High	<ul style="list-style-type: none"> ➤ From the PathPilot interface, on the Status tab, examine the axes' Limits LEDs. If one or more LEDs are on, do the following: <ol style="list-style-type: none"> 1. Identify which axis is stuck on a limit switch. 2. From the PathPilot interface, on the Settings tab, clear the Limit Switches checkbox. Then, on the dialog box, select OK. 3. Jog the axis away from the limit switch that it's on. 4. From the PathPilot interface, on the Settings tab, select the Limit Switches checkbox to re-enable the limit switches. 	If an axis is moved before referencing the machine, it can trigger a limit switch and become stuck. When you disable the limit switches, 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.

Cause: A limit switch is defective.

Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> Go to "The machine is stuck on a limit switch." earlier in this section. Examine each limit switch for its red LED. If any of the limit switch's red LEDs is off, go to "A limit switch's cable or connector is defective." later in this section. 	Each limit switch has a red LED that illuminates whenever the machine is powered on.

Cause: A limit switch's cable or connector is defective.

Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> Power off the machine (see "Power off the Machine" (page 52)). Inspect and reseat the limit switch's connectors on both ends of the cable. Inspect the limit switch's cable for wear or damage. 	Each limit switch's cable is routed from the machine control board to the limit switch, and it has a connector on each end.

Cause: A limit switch flag is improperly adjusted.

Probability	How-To Steps	Need More?
Low	<p>If the machine crashes into the hard stop during the referencing procedure:</p> <ol style="list-style-type: none"> Jog the machine to its reference position. From the PathPilot interface, on the Settings tab, clear the Limit Switches checkbox. Then, on the dialog box, select OK. Push in the Emergency Stop button on the operator box. Adjust the limit switch flag so that the limit switch's red LED is off. Jog the machine off of the limit switch. From the PathPilot interface, on the Settings tab, select the Limit Switches checkbox to re-enable the limit switches. <p>For information, see "The machine is stuck on a limit switch." earlier in this section.</p>	If the limit switch flags are improperly adjusted, the limit switch won't trigger when the machine is referenced. Instead, the axis will crash into the hard stop.

Cause: The control board is defective.

Probability	How-To Steps	Need More?
Low	<p>➤ Go to Limit Switch Function Reference.</p>	A defective control board will report no change in the state of the limit switch, even though the switch and wiring are functioning properly.

12.7.5 Lost Motion on Axis Travel

The machine uses stepper motors — open-loop control motors that are accurate and reliable — to control axis motion. With stepper motors, however, there's a chance of losing steps in axis motion. This is because lost steps occur when the commanded number of steps and the actual number of steps don't match (a risk with open-loop control). A step mismatch results in a loss of motion on the axis. In most cases, when a machine loses steps, it loses many steps all at once — resulting in a visible stutter or a stall in axis motion, and/or an audible noise. Lost steps often occur when a stepper motor is pushed too hard or too fast, and it exceeds its limits.

12: TROUBLESHOOTING

12.7 Axes Drive Subsystem

Although this machine uses stepper motors to control axis motion, the entire system is designed to reduce the likelihood of losing steps. In most cases, the machine breaks smaller cutting tools or stalls the spindle with bigger cutting tools before losing steps. Outside variables, like programming, tooling, workholding, and operator error, are sometimes misinterpreted as lost steps.

Cause: Improper use of tool offset, work offset, or cutter compensation.

Probability	How-To Steps	Need More?
High	<ul style="list-style-type: none"> ➤ Examine the G-code programs. You must fully understand tool offsets, work offsets, and cutter compensation. 	The most common cause of a perceived loss of position or lost steps is operator error.

Cause: The spindle tooling isn't properly clamped (Z-axis only).

Probability	How-To Steps	Need More?
High	<ul style="list-style-type: none"> ➤ Examine the cutter to verify that it's not slipping in the holder, or that the tool holder isn't pulling out of the spindle collet. 	

Cause: The motor coupling is loose or cracked.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Examine the motor coupling. 	You may find it useful to carefully run the axis with the cover removed. Make a paint line from the shaft through the coupling to the screw to examine if there's any movement over time.

Cause: Controller or PathPilot problem.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Restart the controller and send the log file (from the logfiles directory) to Tormach Technical Support. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed. 	For information, see "Getting Help" (page 181).

Cause: There's an obstruction or excessive friction (the gibs aren't adjusted properly, or poor lubrication), or there's high load in the mechanical system.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Jog the axis and carefully observe the motion. 	<p>If you lose steps, it's normally many steps. You should be able to hear the motor. Mechanical issues most often result in losing a large number of steps or stalling. Typical mechanical issues include: an increase in friction due to lack of oil at the way surfaces or ball screw and/or improperly adjusted gibs. They also come from excessive load on the system due to chips or debris on the way surfaces or ball screw, a sticking Z-axis brake or an end-of-travel bumper wedged against the motor mount casting. This occurs sometimes after a limit switch failure and is more common on the Z-axis.</p>

Cause: The axes drivers have the wrong DIP switch settings.

Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ See the machine's electrical schematic. 	<p>New axis drivers require you to set the DIP switches at installation.</p>

12: TROUBLESHOOTING

12.8 Spindle Drive Subsystem

12.8 SPINDLE DRIVE SUBSYSTEM

The machine's spindle is driven by an AC motor whose speed is controlled by a variable frequency drive (VFD).

The spindle is in a ready-to-run condition when:

1. The control power is on.
2. The machine is reset.
3. The spindle brake resistor thermal switch isn't tripped.

To troubleshoot the spindle drive subsystem, read the following:

12.8.1 Spindle Turns in Reverse	196
12.8.2 The Spindle Won't Turn	197

12.8.1 Spindle Turns in Reverse

Cause: The tooling is incorrectly installed.

Probability	How-To Steps
Medium	➤ Check the direction in which the tooling is installed. Inserts should face away from you when installed on the reverse-action quick-change tool post.

Cause: There's no power to the VFD.

Probability	How-To Steps
Low	<ol style="list-style-type: none">1. Power off the machine and the PathPilot controller.<ol style="list-style-type: none">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.d. Follow correct lockout/tagout procedures.2. Remove the VFD's cover with a Phillips screwdriver.3. Identify motor wires #401 and #402 (on the bottom of the VFD). Loosen the set screws above wires #401 and #402 and swap the wire connections. Then, retighten the set screws.4. Put the VFD cover back, and close the electrical cabinet door.5. Power on the machine and the PathPilot controller.<ol style="list-style-type: none">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 machine's Reset button (next to the Emergency Stop button).d. Bring the machine out of reset and reference it.6. From the PathPilot interface, on the Main tab, select Spindle Fwd.7. Confirm the spindle is rotating in the correct forward direction.

12.8.2 The Spindle Won't Turn

Cause: The motor runs, but the spindle doesn't turn.		
Probability	How-To Steps	Need More?
Low	<ul style="list-style-type: none"> ➤ Examine the spindle belt for wear or damage. 	If the display on the VFD is on, the belt may be loose or broken. You'll hear the spindle motor running.
Cause: There's no power to the VFD.		
Probability	How-To Steps	Need More?
—	<ul style="list-style-type: none"> ➤ Examine the VFD display: it has power if its digital lights are on. 	When power is removed, the VFD display remains active until the internal capacitors dissipate their energy. That usually takes about five seconds.

12: TROUBLESHOOTING

12.8 Spindle Drive Subsystem

Cause: There's no power to the VFD because contactor K2 is not energizing. (Examine the voltage across 112/N and 117 at the VFD, which should be 115 Vac nominal.)

Probability	How-To Steps
High	<p>There are loose power or control wires in the VFD circuit.</p> <ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 2. Examine the circuit for loose wires. 3. Power on the machine (see "Power on the Machine" (page 49)) and test operation.
Probability	How-To Steps
Medium	<ol style="list-style-type: none"> 1. Examine the spindle cover door to verify that it's being held closed. 2. Verify that there's 24 Vdc measured from wire 400 to wire 436.
Probability	How-To Steps
Low	<p>Thermal switch (TS1) tripped, preventing K2 from latching.</p> <ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 2. Allow the brake resistor to cool, and reset thermal switch by pressing reset button (between its two terminals).
Probability	How-To Steps
Low	<p>The control board isn't providing a run command or holding the K2 contactor on.</p> <ol style="list-style-type: none"> 1. Examine wires 434 and 436 for 24 Vdc on wires J10.1 and J10.3, respectively. 2. Start the spindle and listen for a soft, audible click on the control board. If you hear this click (from a relay contact on the board), the machine control board is functioning properly. If you don't hear the click: <ul style="list-style-type: none"> • Verify that there's 24 Vdc measured from wire 400 to wire 434. Make a jumper wire and, carefully, momentarily jumper wires 434 and 436. <ul style="list-style-type: none"> ◦ If contactor K2 pulls in (you will hear an audible clunk) while you have the jumper on but drops out as soon as you remove the jumper, the holding contact on K2 is defective. ◦ If K2 stays powered on, the control board is not passing the run signal to the circuit. The control board passes 24 Vdc from wire 434 to 436 via a relay to create the start pulse. Measure wire 434 for 24 Vdc power. If present, the control board or wire 436 connected to J10.3 is defective. Power off the machine (see "Power off the Machine" (page 52)), and jumper J10.1 to J10.3. Power on the machine (see "Power on the Machine" (page 49)) and check the VFD for a display. If the VFD reads rdy, the control board is defective. If not, wire 436 may be broken. Lift the connections of wire 436 at the control board and K2 and measure continuity.

Cause: The VFD tripped.

Probability	How-To Steps	Need More?
Low	<p>➤ If the VFD tripped, an error code displays. Read the error code and go to "Spindle VFD Trip Reference" (page 200).</p>	<p>You can clear a VFD trip by either:</p> <ul style="list-style-type: none"> • Removing power from the VFD for 30 seconds. • Pressing the red Reset button on the front of the VFD .

Cause: The VFD is defective.

Probability	How-To Steps
Low	The VFD may be defective if: <ul style="list-style-type: none"> • The display isn't on and there is nominal 115 Vac between wires 117 and 112/N at the VFD. • The VFD displays a trip condition that does not clear when power is removed, the VFD may be defective.

Cause: The VFD is not programmed, or it's programmed incorrectly.

Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> 1. Push Enter on the front panel of the VFD twice. The display changes to 00.000, with the .000 blinking. 2. Repeatedly push the Up Arrow until .000 changes to .011. 3. Push Enter again. The display shows the model of the machine or accessory (for example, 1100 for an 1100M, or RT11 for a RapidTurn on an 1100M). 4. Push the Up Arrow once more, and the VFD displays parameter .012, which is the software version number (for example, version 2.01). 5. Push Back to exit this mode. 	Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.

Cause: The machine control board is defective, or there are defective cables between the machine control board and the spindle VFD.

Probability	How-To Steps
Low	<ul style="list-style-type: none"> ➤ Examine all cables to verify that they're properly seated in their connectors on the machine control board.

Cause: The motor is defective.

Probability	How-To Steps	Need More?
Low	<ol style="list-style-type: none"> 1. Power off the machine (see "Power off the Machine" (page 52)). 2. Wait 30 seconds, and then remove wires 118, 119, and 120 from the VFD terminals. 3. Measure the resistance between: <ul style="list-style-type: none"> • Wires 118 and 119 • Wires 119 and 120 • Wires 120 and 118 Resistance should be in the range of approximately 2-4 Ω. 	<ul style="list-style-type: none"> • 0 Ω indicates that the winding is shorted. • >1M Ω indicates that the winding is open. Both cases indicate a defective motor or compromised wiring to the motor from the VFD.

Run and Direction Commands Reference

Command From Card	Monitoring Points One Probe on Each		Voltage Measured	
	Common wire number	Wire number	Voltage when control board command is on	Voltage when control board command is not on
Run (FWD)	403	404	14-20 Vdc	0 Vdc
Reverse	403	406	14-20 Vdc	0 Vdc

12: TROUBLESHOOTING

12.8 Spindle Drive Subsystem

The display on the VFD provides valuable information for troubleshooting. The display diagnostics include:

- Frequency output (proportional to speed. Range is ~7 Hz to 142 Hz).
- Status (rd for ready, inh for inhibit which will occur when there is no jumper between terminals B2 and B4 on the drive).
- Fault information (Er for trip) and a code for the fault.

Spindle VFD Trip Reference

Trip Code	Condition	Likely Cause
UU	DC-BUS under-voltage.	This happens when the VFD is powered down.
OU	DC-BUS over-voltage.	Braking resistor failed open or wiring connection open between the VFD and the resistor. Resistance to measure 70 ohms.
OI.AC	VFD output instantaneous over current.	Phase to phase or phase to ground short on output of VFD to motor. This trip code cannot be reset until 10 seconds after the trip was initiated.
OI.br	Braking resistor instantaneous over current.	Braking resistor shorted or partially shorted out or short in wiring between the VFD and the resistor. Resistance to measure 70 ohms. Check brake resistor wiring.
It.br	I^2t (power) on braking resistor.	Excessive braking resistor energy caused by too frequent and too severe deceleration cycles or AC supply voltage too high.
It.AC	I^2t (power) on VFD output current (used to protect motor).	You are working the spindle motor too hard. Ensure that the spindle is not jammed or sticking. Consider running the spindle motor at half speed for 10 minutes with no load to cool the motor down.
Oht.C	VFD is working too hard and stops to cool power electronics down to prevent failure.	Spindle motor working too hard. Stop running the spindle but leave the VFD power on and let the power electronics cool down.
Oht.I	Heat sink temperature is too high because the VFD is working too hard and stops to cool power electronics down to prevent failure. Cabinet may also be too hot.	Spindle motor working too hard or it is too hot in work location. Stop running the spindle but leave the VFD power on and let the power electronics cool down. Check to see if the fan on the VFD is running and check filters on the cabinet. Cool work location down if required.
HF01 through HF23	Cooling fan is not cooling.	Failed drive.

12.9 OPERATOR CONSOLE TROUBLESHOOTING

12.9.1 The Screen Doesn't Respond to Touch Inputs	201
12.9.2 The Screen Doesn't Display an Image or Respond to Power Button	202
12.9.3 The Screen is Scrambled or Illegible	202
12.9.4 The Knobs Don't Respond	203
12.9.5 The Buttons Don't Respond	204

12.9.1 The Screen Doesn't Respond to Touch Inputs

Problem

The touch screen does not respond to touch inputs on all or part of the screen's surface.

Cause

The sensitivity setting for the touch controller is too low.

Solutions

You Might Need To...	Probability	How-To Steps	Need More?
Adjust touchscreen sensitivity.	High	<ol style="list-style-type: none"> 1. Verify that you have PathPilot v2.4.4 or higher installed on your controller. 2. From the PathPilot interface, in the MDI Line DRO field, type <code>ADMIN TOUCHSCREEN SENSITIVITY</code> <code>1000</code> and press Enter. You can use a value between 1 and 2047, but 1000 is generally sufficient for most shop spaces. 3. Verify that the touch screen responds to touch inputs. If it doesn't, go to the next step. 4. From the PathPilot interface, on the File tab, find the <code>pointercal.xinput</code> file and delete it. 5. Restart the PathPilot controller. The calibration utility displays. For now, skip this procedure. 6. From the PathPilot interface, in the MDI Line DRO field, type <code>ADMIN TOUCHSCREEN SENSITIVITY</code> <code>1000</code> and press Enter. 7. From the PathPilot interface, in the MDI Line DRO field, type <code>ADMIN TOUCHSCREEN</code> and press Enter. The calibration utility displays. Use your finger (not a mouse) to touch all four points that display on the screen. 	The touchscreen is a resistive type to prevent accidental triggering from drops of coolant on the screen. The resistive touchscreen may need its sensitivity adjusted when used in a shop space with very high or low humidity.

12: TROUBLESHOOTING

12.9 Operator Console Troubleshooting

12.9.2 The Screen Doesn't Display an Image or Respond to Power Button

Problem

The console screen doesn't display an image or respond to the power button.

Cause

The console isn't receiving power.

Solutions

You Might Need To...	Probability	How-To Steps	Need More?
Examine power input to the console.	High	➤ Examine the green LED on the power brick for the console. If it's not lit, examine the power cords to the power brick.	If your console receives power from the Accessory Input ports on the machine, look for tripped breakers inside your machine's electrical cabinet.
Test the power button functionality.	Low	➤ Examine the green ring around the power button. It should light up when you press the power button.	

12.9.3 The Screen is Scrambled or Illegible

Problem

The console screen turns on, but is scrambled or illegible.

Cause

The BIOS isn't configured for the correct screen output.

Solutions

You Might Need To...	Probability	How-To Steps	Need More?
Configure the display output settings in BIOS.	High	<ol style="list-style-type: none"> 1. Connect a VGA monitor to the console. 2. Power the console on and select the Delete key to enter the BIOS. 3. From the Advanced tab, select Display Configuration. 4. Configure the display as follows: <ul style="list-style-type: none"> • Primary IGFX Boot Display: Auto • LCD Panel Type: 1280x1024 LVDS • Panel Channel: Dual Channel • Panel Color Depth: 24 Bit 5. Select the Esc key, go to Save and Exit, and select Save Changes and Reset. 	This configuration problem can occur if your console has a CMOS battery failure. Replace the battery if it reoccurs.

12.9.4 The Knobs Don't Respond

Problem

The RPM, Feed Override, or Max Velocity knobs don't respond or aren't smooth.

Cause

The ribbon cable connecting the knobs is disconnected or the circuit board is damaged.

Solutions

You Might Need To...	Probability	How-To Steps	Need More?
Examine the connectors on the ribbon cable.	High	<ol style="list-style-type: none"> 1. Remove the rear panel of the console. 2. Examine the connectors on both ends of the cable going from J4 on the control board to the potentiometer board. 	Shipping can sometime cause connectors to become loose. Re-seating the connectors will usually fix non-responsive override knobs.

12: TROUBLESHOOTING

12.9 Operator Console Troubleshooting

You Might Need To...	Probability	How-To Steps	Need More?
Examine the USB connection to the control board.	High	<ol style="list-style-type: none"> 1. Remove the rear panel of the console. 2. Examine the USB cable going from the header on the computer motherboard to connector J12 on the control board. 	Verify that the power LED on the console control board lights up when the console is turned on. If it doesn't light up, and you have confirmed the USB connection, replace the control board (PN 39146).

12.9.5 The Buttons Don't Respond

Problem

The Cycle Start or Feed Hold buttons don't respond.

Cause

The control board is disconnected or the wires to the buttons are loose.

Solutions

You Might Need To...	Probability	How-To Steps	Need More?
Examine the wiring to the buttons.	High	<ol style="list-style-type: none"> 1. Remove the rear panel of the console. 2. Examine the wire inputs to connector J13 on the control board. If any wires are loose, tighten the screw terminals. 3. Using a continuity tester, measure the resistance between terminals 1 and 2 when Feed Hold is pressed and 3 and 4 when Cycle Start is pressed. <ul style="list-style-type: none"> • If there's continuity at the terminals on the control board and the buttons still don't work, examine the USB cable to the control board. • If there's not continuity at the terminals when the buttons are pressed, remove the lower rear panel of the console and examine the screw terminals on the rear of the buttons themselves. 	Shipping can sometime cause wire terminals to become loose. Re-seating the wires will usually fix non-responsive buttons. If you have tested all terminals and the buttons still don't have continuity when pressed, replace the buttons: <ul style="list-style-type: none"> • Feed Hold Button (PN 37363) • Cycle Start Button (PN 37362)
Examine the USB connection to the control board.	High	<ol style="list-style-type: none"> 1. Remove the rear panel of the console. 2. Examine the USB cable going from the header on the computer motherboard to connector J12 on the control board. 	Verify that the power LED on the console control board lights up when the console is turned on. If it doesn't light up and you have confirmed the USB connection, replace the control board (PN 39146).

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.

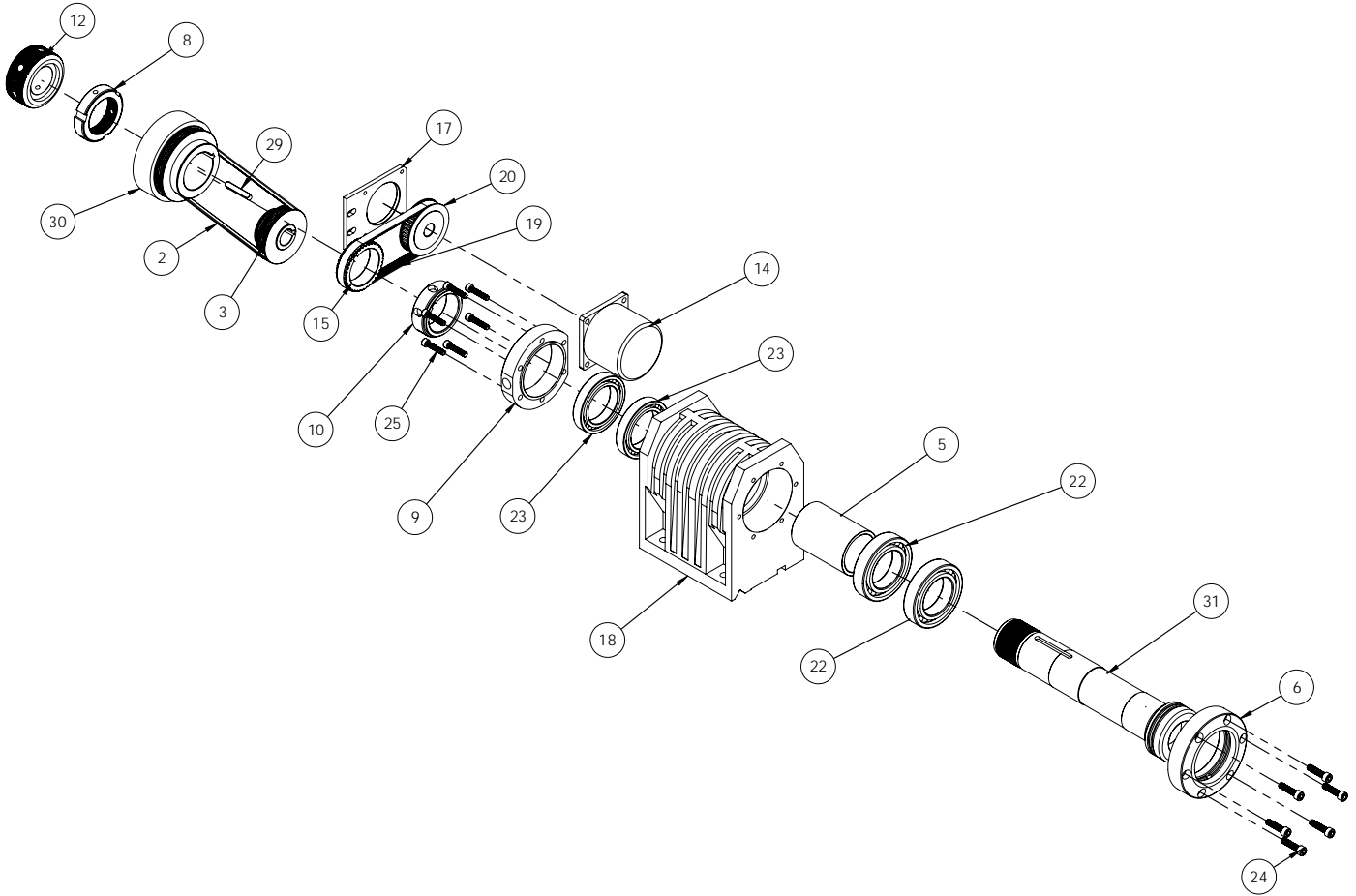
CONTENTS

13.1 Headstock	206
13.2 Stand	208
13.3 Tailstock	210
13.4 X-Axis	212
13.5 Z-Axis	215
13.6 Electrical Cabinet	217

13: DIAGRAMS AND PARTS LISTS

13.1 Headstock

13.1 HEADSTOCK

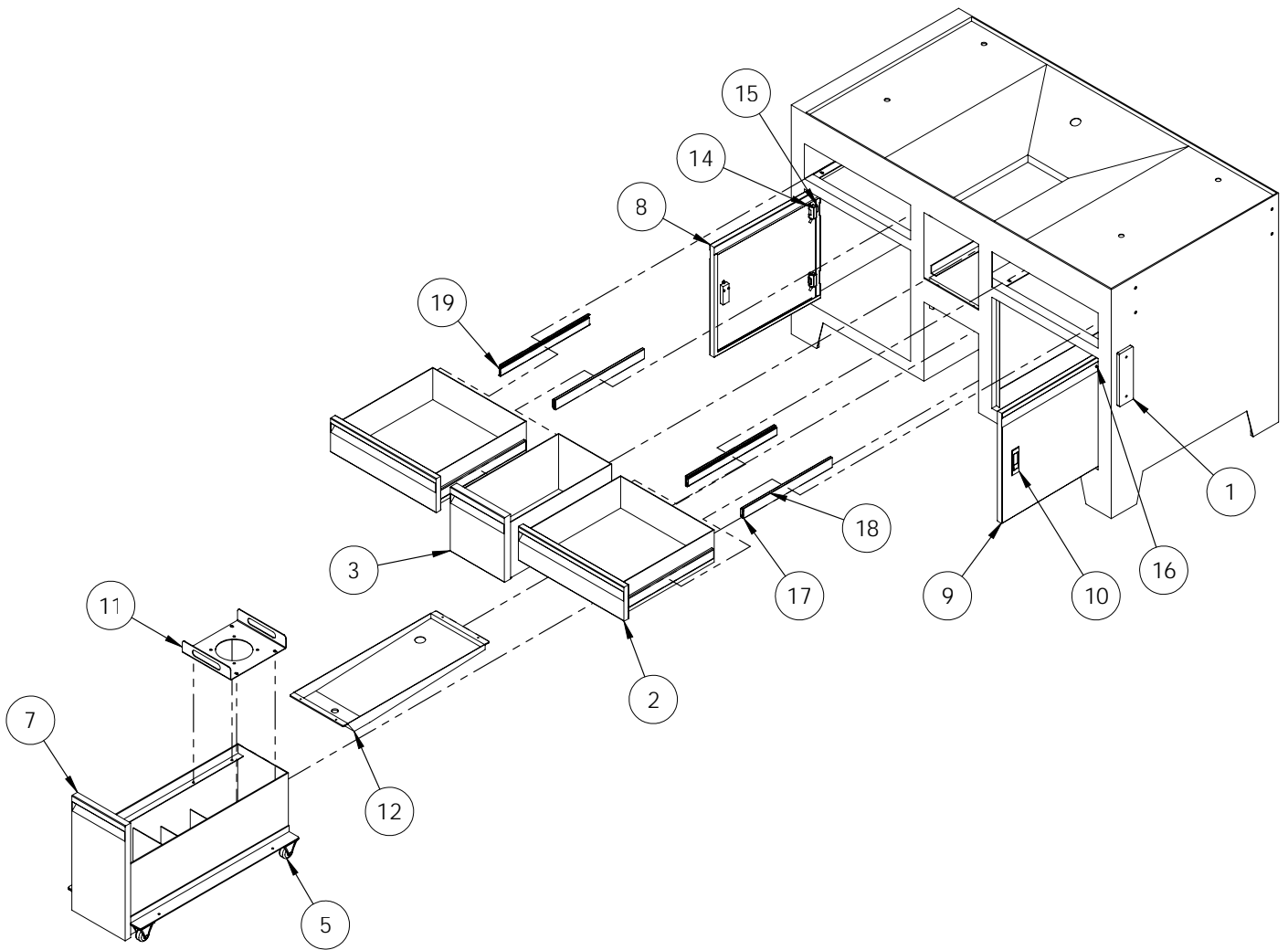


ID	Description	Quantity
2	Spindle Belt, 8L (PN 37832)Spindle Belt, 8L	1
3	Spindle Motor Pulley (PN 37830)	1
4	Spindle (PN 37837)	1
5	Spacer (PN 37838)	1
6	Spindle Front Retainer (PN 37835)	1
7	Spacer (PN 37843)	1
8	Spindle Retainer (PN 37844)	1
9	Spindle Rear Retainer (PN 37836)	1
10	Spindle Lock Ring (PN 37845)	1
11	Spacer (PN 37842)	1
12	Draw Tube Knob (PN 37841)	1
13	Collet Draw Tube (PN 37839)	1
14	Spindle Encoder (PN 37826)	1
15	Encoder Spindle Shaft Pulley (PN 37840)	1
16	Spindle Pulley (PN 37831)	1
17	Spindle Encoder Mounting Plate (PN 37829)	1
18	Headstock Frame (PN 37824)	1
19	Spindle Encoder Belt (PN 37833)	1
20	Encoder Pulley (PN 37834)	1
21	8L Drawtube Assembly (PN 51892)	1
22	Rolling Bearings 6009 (PN 51043)	2
23	Rolling Bearings 61909 (PN 51044)	2
24	M6 × 25 Screw (PN 51039)	6
25	M5 × 25 Screw (PN 51045)	6
26	M5 × 5 Screw (PN 51046)	2
27	M6 × 8 Screw (PN 51047)	3
28	M6 × 6 Screw (PN 51048)	3
29	6 × 36 Key (PN 51049)	1
30	Spindle Pulley (PN 37853)	1
31	Spindle (PN 37852)	1

13: DIAGRAMS AND PARTS LISTS

13.2 Stand

13.2 STAND

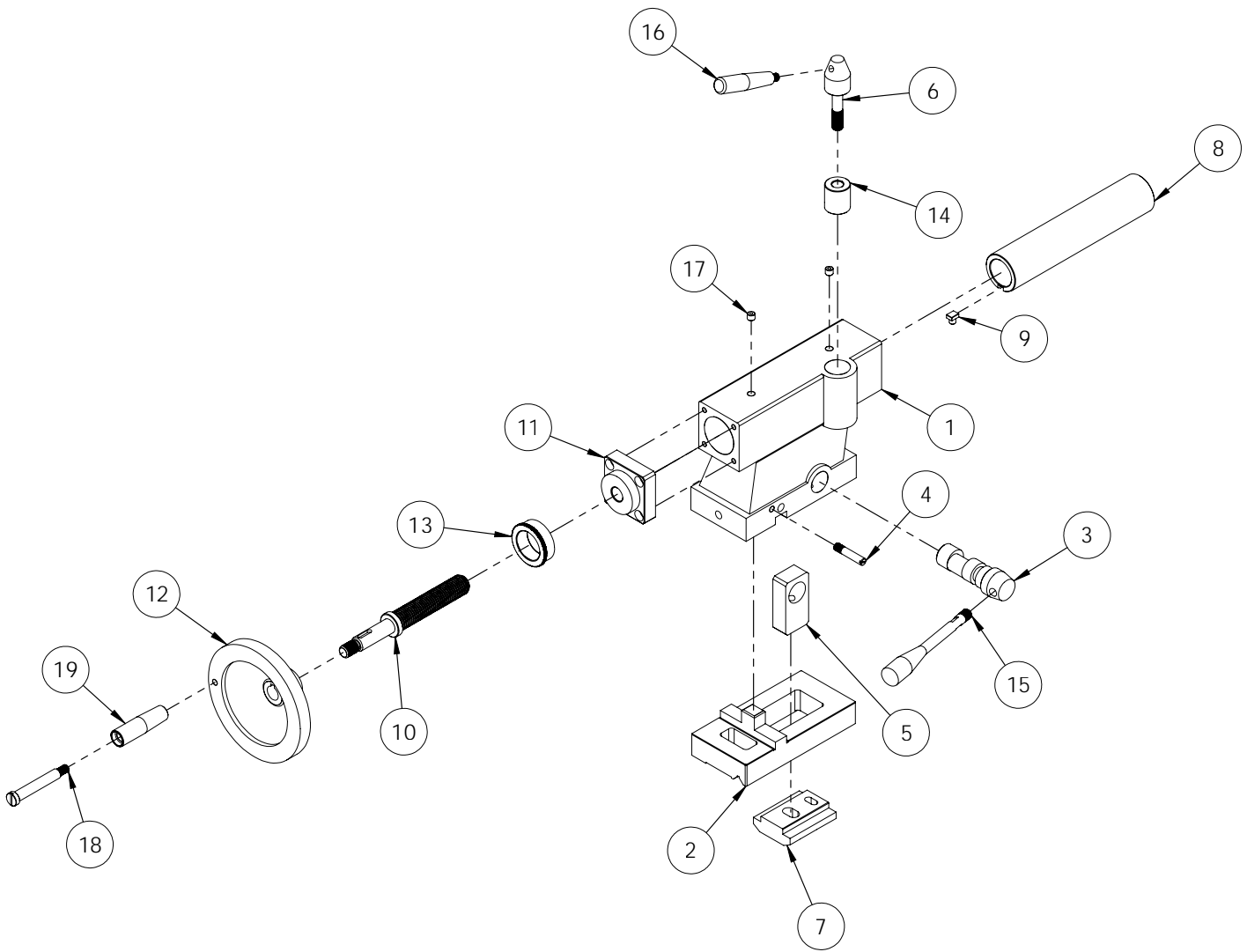


ID	Description	Quantity
1	Machine Stand Frame (PN 50553)	1
2	Machine Stand Drawer (PN 50554)	2
3	Machine Stand Chip Drawer (PN 50555)	1
4	Coolant Tank Plug Bung (PN 50566)	1
5	Coolant Tank Caster (PN 50567)	4
6	Coolant Tank Plug (PN 50565)	1
7	Coolant Tank (PN 50563)	1
8	Left Stand Door (PN 50557)	1
9	Right Stand Door (PN 50558)	1
10	Door Latch (PN 50561)	2
11	Coolant Pump Flange (PN 50564)	1
12	Coolant Funnel in Stand (PN 50559)	1
13	Stand Shelf (PN 50556)	2
14	Door Hinge (PN 505601)	4
15	Stand Hinge (PN 505602)	4
16	Hinge Pin (PN 505603)	4
17	Drawer Slide Out (PN 505621)	4
18	Drawer Slide Inner (PN 505622)	4
19	Drawer Slide Bearing (PN 505623)	4

13: DIAGRAMS AND PARTS LISTS

13.3 Tailstock

13.3 TAILSTOCK

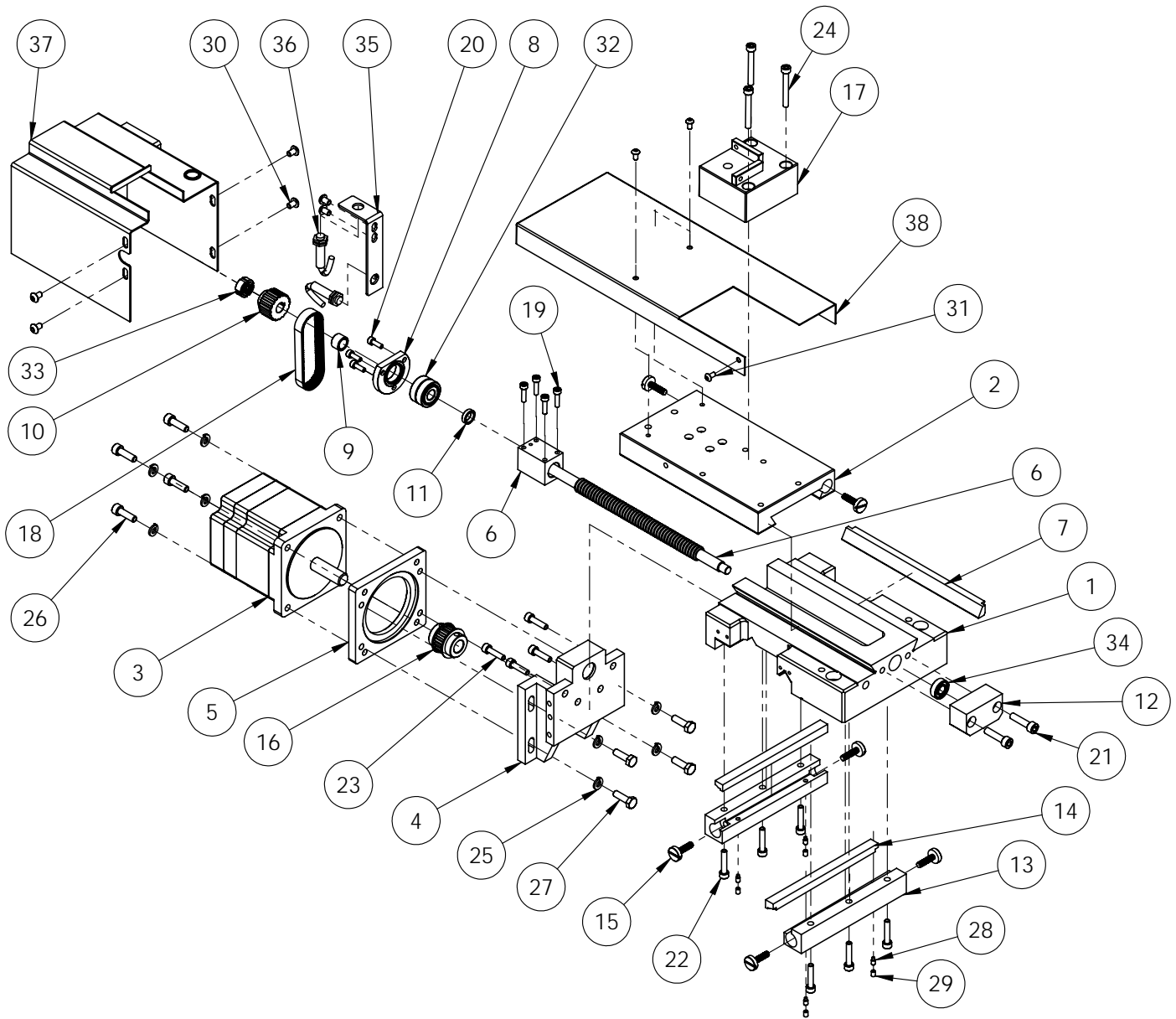


ID	Description	Quantity
1	Tailstock Frame (PN 37805)	1
2	Tailstock Base (PN 37806)	1
3	Tailstock Location Cam (PN 37821)	1
4	Fastner (PN 37823)	1
5	Tailstock Lock (PN 37822)	1
6	Tailstock Slide Lock (PN 37816)	1
7	Tailstock Location Lock Block (PN 37819)	1
8	MT2 Adapter (PN 37807)	1
9	Slide Guide (PN 37818)	1
10	Tailstock Slide Screw (PN 37811)	1
11	Tailstock Slide Screw Nut (PN 37808)	1
12	Hand Wheel (PN 37809)	1
13	Graduation Wheel (PN 37810)	1
14	Slide Lock Mount (PN 37817)	1
15	Tailstock Location Lock Handle (PN 37820)	1
16	Tailstock Handle (PN 37815)	1
17	Spring Plunger (PN 37814)	2
18	Shoulder Screw (PN 37813)	1
19	Tailstock Wheel Handle (PN 37812)	1

13: DIAGRAMS AND PARTS LISTS

13.4 X-Axis

13.4 X-AXIS



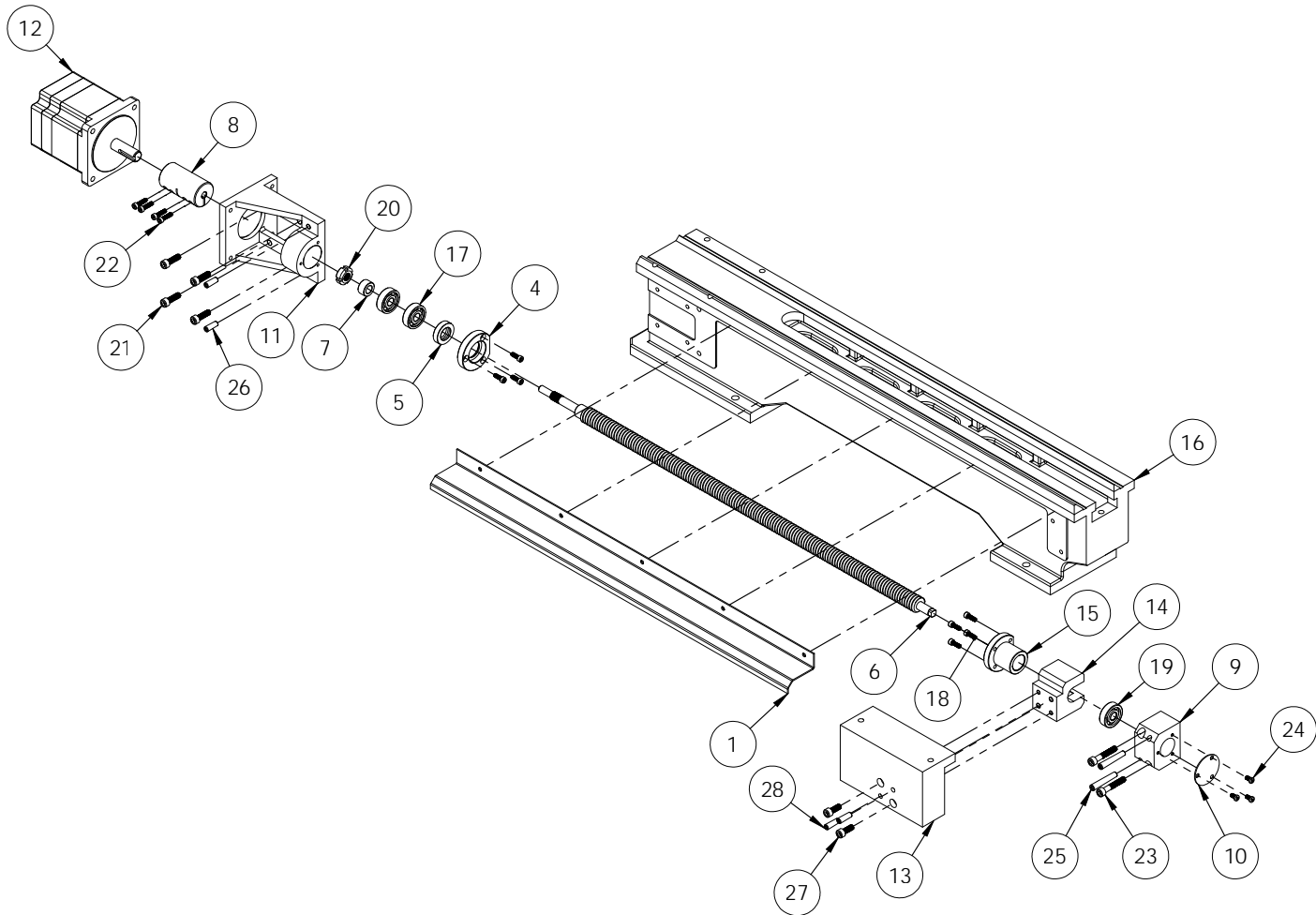
ID	Description	Quantity
1	X-Axis Base (PN 37780)	1
2	X-Axis Slide (PN 37781)	1
3	Stepper Motor (PN 39533)	1
4	X-Axis Motor Mount (PN 37782)	1
5	X-Axis Motor Adjustment Plate (PN 37784)	1
6	Ball Screw Assembly, 14 mm OD, 4 mm Pitch, 160 mm Threaded Length (PN 53414)	1
	X-Axis Ball Screw, 160 mm Long (PN 37792)	1
	X-Axis Ball Screw Nut (PN 37793)	1
7	X-Axis Gib (PN 37787)	1
8	X-Axis Ball Screw Retainer (PN 37788)	1
9	Spacer, 7.5 THK, 10 ID × 14.5 OD (PN 37797)	1
10	X-Axis Ball Screw Pulley (PN 37790)	1
11	Spacer, 3.5 THK, 10 ID × 14 OD (PN 37795)	1
12	X-Axis Ball Screw Support (PN 37794)	1
13	Gib Retainer, Z-Axis, 8L Lathe (PN 37785)	2
14	Gib, Z-Axis, 8L Lathe (PN 37786)	2
15	Gib Adjustment Screw, Z-Axis, 8L Lathe (PN 37799)	6
16	X-Axis Motor Pulley (PN 37849)	1
17	Quick Change Tool Post Riser (PN 50603)	1
18	X-Axis Motor Belt, 8L (PN 53687)	1
19	Screw, Socket Head Cap, M4 × 0.7 - 16 (PN 37751)	4
20	Screw, Socket Head Cap, M4 × 0.7 - 12 (PN 31360)	3
21	Screw, Socket Head Cap, M6 × 1 - 25 (PN 31685)	2
22	Screw, Socket Head Cap, M5 × 0.8 - 25 (PN 30530)	6
23	Screw, Socket Head Cap, M5 × 0.8 - 20 (PN 30357)	4
24	Screw, Socket Head Cap, M5 × 0.8 - 40 (PN 52432)	3
25	Washer, Split Lock, M6 (PN 31379)	8
26	Screw, Socket Head Cap, M6 × 1 - 20 (PN 30832)	4
27	Screw, Hex Head Cap, M6 × 1 - 20 (PN 54793)	4
28	Screw, Extended Tip Set, M4 × 0.7 - 8, (2.25 mm × 2.5 mm Dia Tip) Black Oxide Steel (PN 54794)	4
29	Screw, Flat Point Set, M4 × 0.7 - 6 (PN 54795) 4	

13: DIAGRAMS AND PARTS LISTS

13.4 X-Axis

ID	Description	Quantity
30	Screw, Button Head Cap, M5 × 0.8 - 8, Stainless Steel (PN 34636)	6
31	Screw, Button Head Cap, M4 × 0.7 - 8, Stainless Steel (PN 50838)	3
32	Bearing, 15 Degree Angular Contact, 7000C, 10 mm ID × 26 mm OD, 8 mm Wide, Sealed, Matched Set of 2 (PN 53685)	2
33	Lock Nut, M10, 8L (PN 51027)	1
34	Bearing, Deep Groove, 698, 8 mm × 19 mm × 6 mm (PN 51037)	1
35	Bracket, X-Axis Motor Cover, 8L (PN 54780)	1
36	N/C Inductive Proximity Sensor (PN 38065)	2
37	X-Axis Motor Cover (PN 50522)	1
38	X-Axis Table Cover (PN 50523)	1

13.5 Z-AXIS

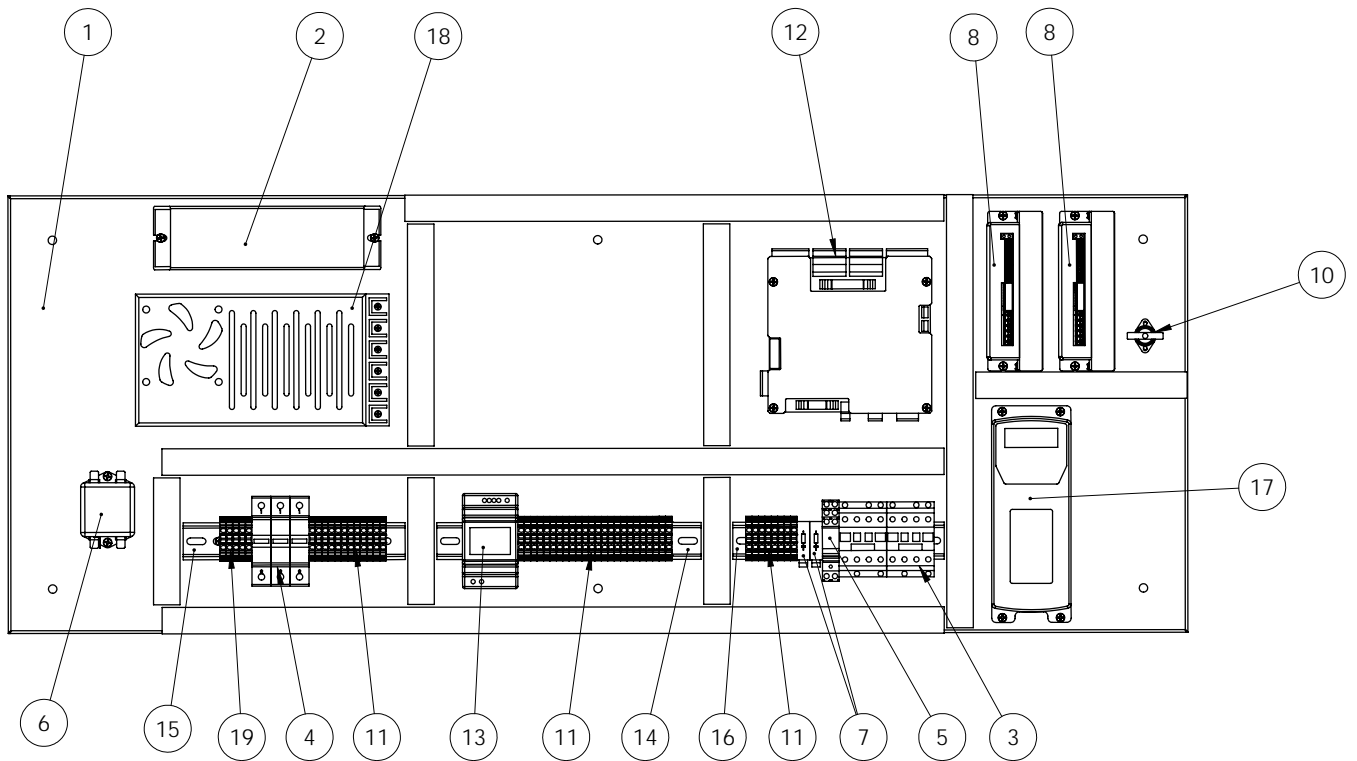


13: DIAGRAMS AND PARTS LISTS

13.5 Z-Axis

ID	Description	Quantity
1	Ball Screw Shield (PN 37851)	1
4	Ball Screw Plate (PN 37775)	1
5	Z-Axis Ball Screw Seal (PN 37772)	1
6	Z-Axis Ball Screw, 555 mm (PN 37771)*	1
7	8L Z-Axis Ball Screw Bearing Spacer (PN 37779)	1
8	Ball Screw Coupler (PN 37773)	1
9	Z-Axis Ball Screw Support (PN 37765)	1
10	Ball Screw Cover (PN 37766)	1
11	Z-Axis Motor Mount (PN 37764)	1
12	Stepper Motor (PN 39533)	1
13	Z-Axis Carriage Mount (PN 37769)	1
14	Z-Axis Ball Screw Block (PN 37768)	1
15	Z-Axis Ball Screw Nut (PN 37770)*	1
16	Z-Axis Way (PN 37763)	1
17	Bearing, 15 Degree Angular Contact, 7201C, 12 mm ID × 32 mm OD, 10 mm Wide, Sealed, Match Set of 2 (PN 51024)	2
18	M4 × 12 Screw (PN 51025)	7
19	Bearing, Ball, 6201RS, 12 mm ID × 32 mm OD × 10 mm Wide, Sealed (PN 51026)	1
20	M10 × 1 Lock Nut (PN 51027)	1
21	M6 × 20 Screw (PN 51028)	4
22	M4 × 16 Screw (PN 51029)	4
23	M6 × 35 Screw (PN 51030)	2
24	M4 × 8 Phillips head screw (PN 51031)	3
25	6 × 40 Taper Pin (PN 51032)	2
26	6 × 18 Taper Pin (PN 51033)	2
27	M6 × 16 Screw (PN 51034)	2
28	6 × 20 Taper Pin (PN 51035)	2
<p>*Ball screws and ball nuts are only sold as an assembly. For replacements, order Z-Axis Ball Screw Assembly, 8L (PN 53415).</p>		

13.6 ELECTRICAL CABINET



13: DIAGRAMS AND PARTS LISTS

13.6 Electrical Cabinet

ID	Description	Quantity
1	Electrical Cabinet Back Plate, 8L (PN 54755)	1
2	Spindle Brake Resistor (PN 37906)	1
3	Contactor Relay, 3-Position, 220 V, 12 A, 24 Vdc Coil, Screw Terminal (PN 37343)	2
4	Circuit Breaker (15A, 1 Pole) (PN 37521)	3
5	Relay, DPDT, 24VDC Coil, 400VAC @ 8A (PN 50830)	1
6	Filter (110 Vac/250 Vac, 20 A) (PN 37914)	1
7	Electrical Noise Suppressor (DIN Mount) (PN 37358)	2
8	<u>Stepper Driver (PN 53488)</u>	2
9	ECM1 Board Shield (PN 37518)	1
10	Thermal Switch, Manual Reset, Ceramic, 220 V, 10 A, 85C (PN 37420)	1
11	Terminal Block, Discrete (PN 32667)	44
12	ECM1v1.5, Machine Control Board (PN 37509)	1
13	Power Supply (24 Vdc, 60 W) (PN 50461)	1
14	DIN Rail (35 × 7.5), 250 mm Long (PN 37910)	1
15	DIN Rail (35 × 7.5), 210 mm Long (PN 37910)	1
16	DIN Rail (35 × 7.5), 200 mm Long (PN 37910)	1
17	VFD (1.5 hp, 120 Vac Input) (PN 35717)	1
18	Power Supply, 90-132 Vac / 180-264 Vac, 75 Vdc, 600 W, 8 A (PN 39101)	1
19	Terminal Block, Discrete, Ground (PN 54761)	5

ELECTRICAL SCHEMATICS

IN THIS SECTION, YOU'LL LEARN:

- About the electrical schematics for this machine's electronics.

The electrical schematic varies by serial number. Identify your machine's serial number (on the right-hand side of the machine, near the Main Disconnect switch). If the serial number is **10506 or higher**, refer to the electrical schematics in this manual. If the serial number is **10505 or lower**, refer to [Tormach document TD10829](#).

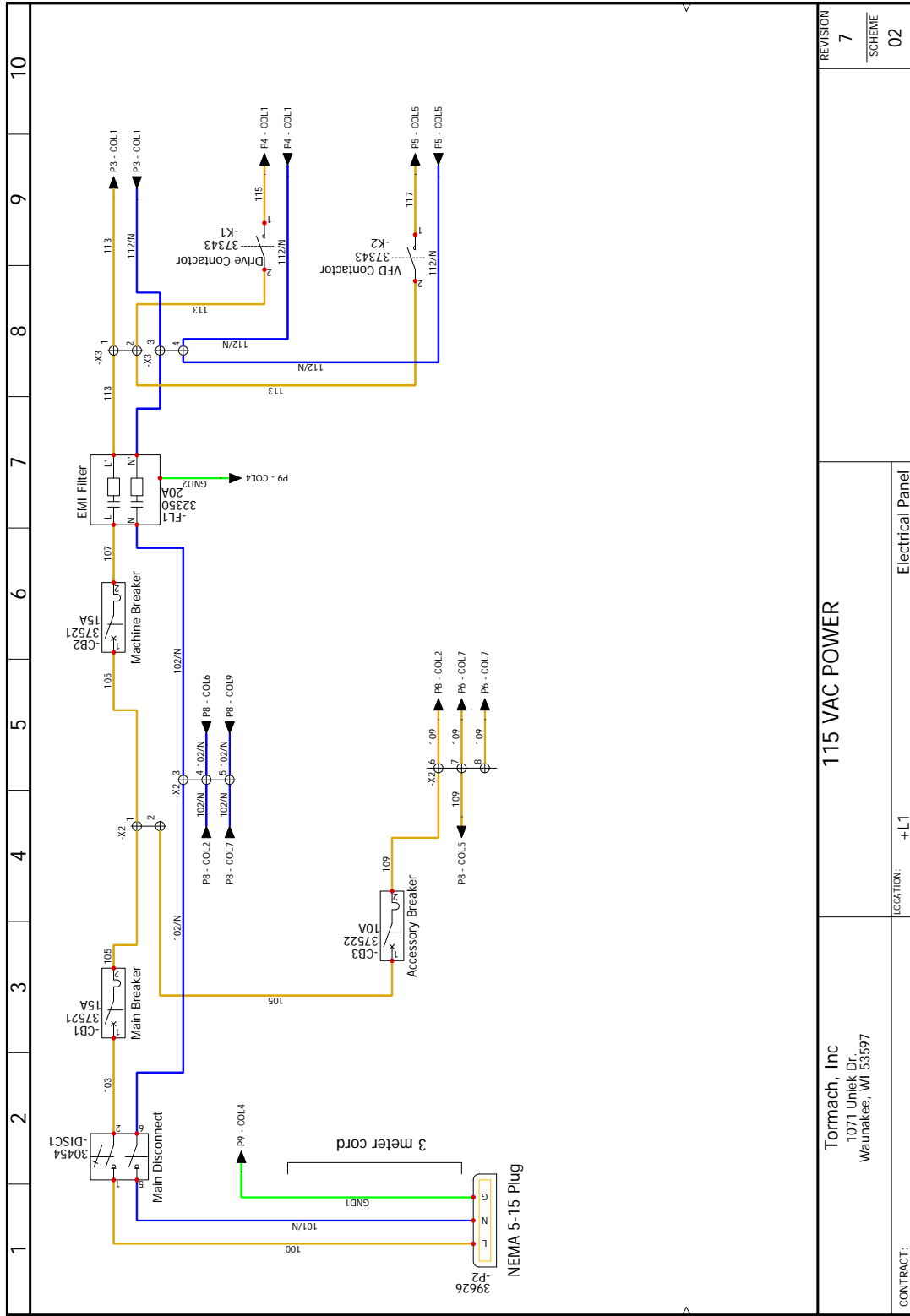
CONTENTS

14.1 115 Vac Power (Sheet 2)	220
14.2 24 Vdc Controls (Sheet 3).....	221
14.3 Axis Drive Bus (Sheet 4)	222
14.4 Spindle Drive (Sheet 5)	223
14.5 Machine Control Board (Sheet 6).....	224
14.6 Limit/Door Switches (Sheet 7).....	225
14.7 Accessory Power (Sheet 8).....	226
14.8 Grounds (Sheet 9).....	227
14.9 Grounds Terminal Strip (Sheet 10).....	228
14.10 AC Terminal Strip (Before EMI1) (Sheet 11).....	229
14.11 AC Terminal Strip (After EMI1) (Sheet 12).....	230
14.12 Control Terminal Strip (Sheets 13-15).....	231
14.13 Limit Switches Terminal Strip (Sheet 16).....	234
14.14 Operator Console Schematic	235

14: ELECTRICAL SCHEMATICS

14.1 115 Vac Power (Sheet 2)

14.1 115 VAC POWER (SHEET 2)



SOLIDWORKS Electrical

115 VAC POWER

Tormach, Inc
1071 Uniek Dr.
Waunakee, WI 53597

CONTRACT:

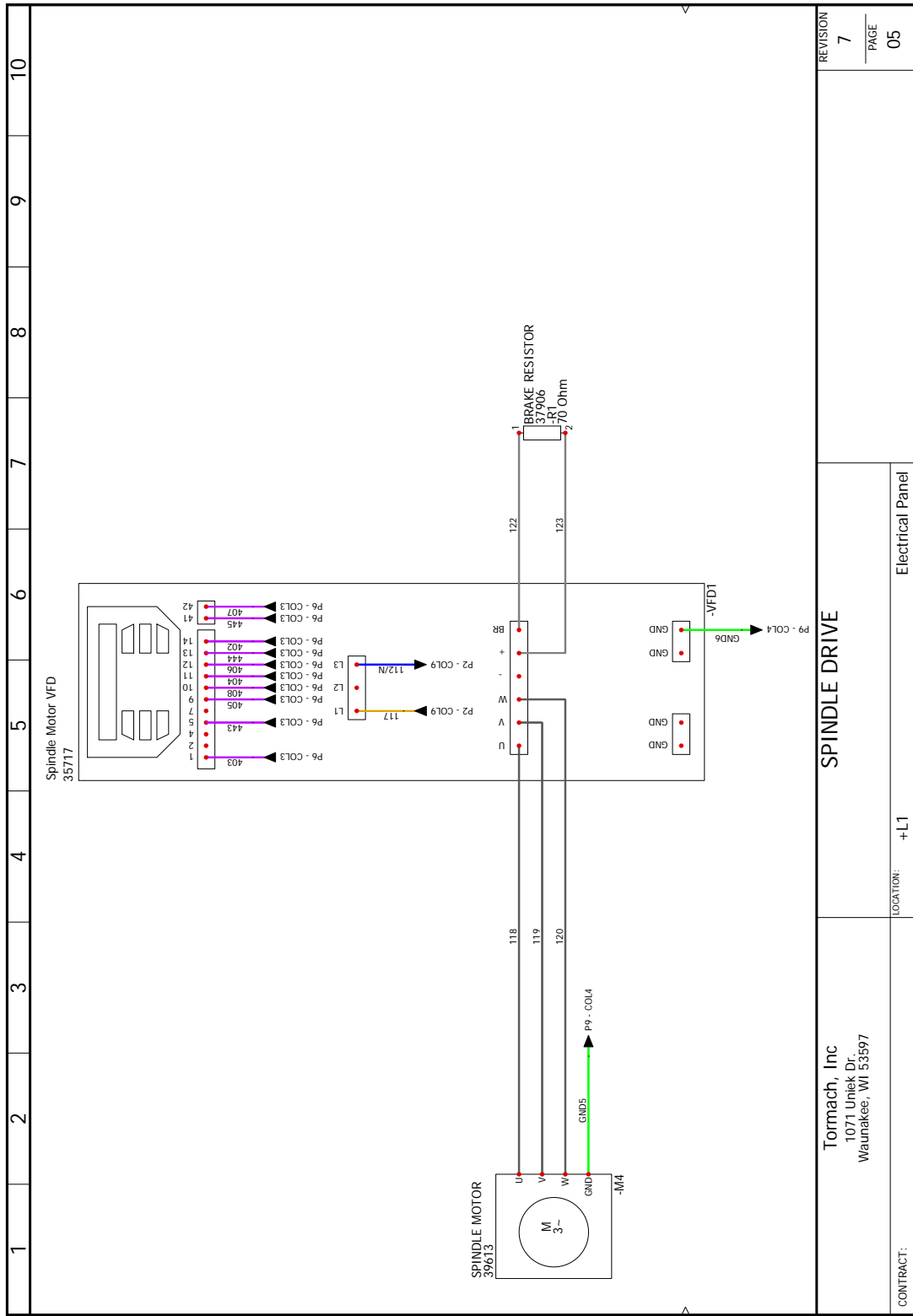
LOCATION: +L1

Electrical Panel

REVISION	7
SCHEME	02

Document released with version: 2025-0-0-50550

14.4 SPINDLE DRIVE (SHEET 5)



SOLIDWORKS Electrical

Tormach, Inc
1071 Uniek Dr.
Waunakee, WI 53597

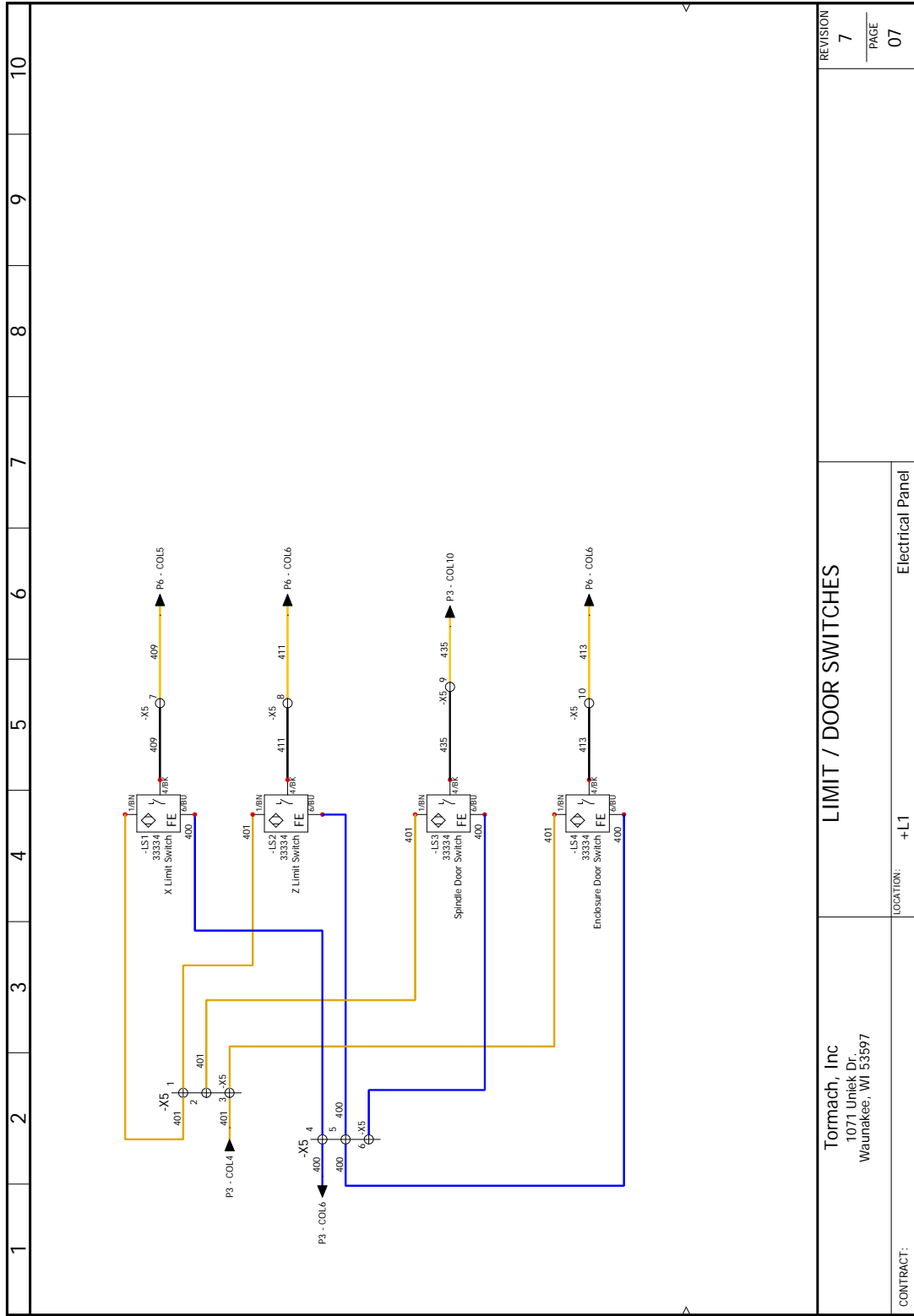
CONTRACT: LOCATION: +L1

SPINDLE DRIVE

REVISION
7
PAGE
05

Document created with version: 2022.1.0. 8/20/20

14.6 LIMIT/DOOR SWITCHES (SHEET 7)



SOLIDWORKS Electrical

REVISION	7
PAGE	07

LIMIT / DOOR SWITCHES

LOCATION: +L1

Electrical Panel

Tormach, Inc
 1071 Uniek Dr.
 Waunakee, WI 53597

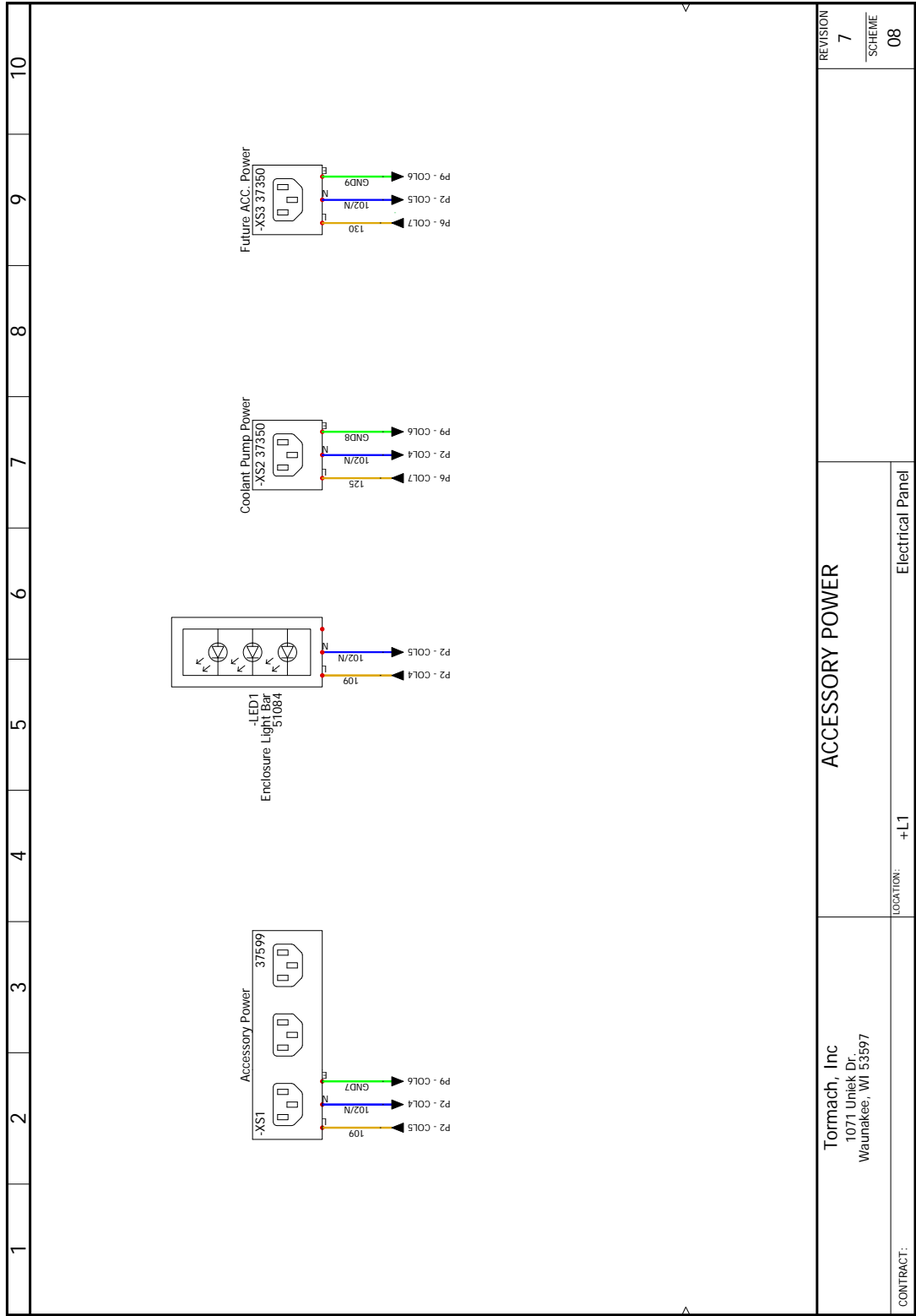
CONTRACT:

Document created with version: 2025.0.0.0000

14: ELECTRICAL SCHEMATICS

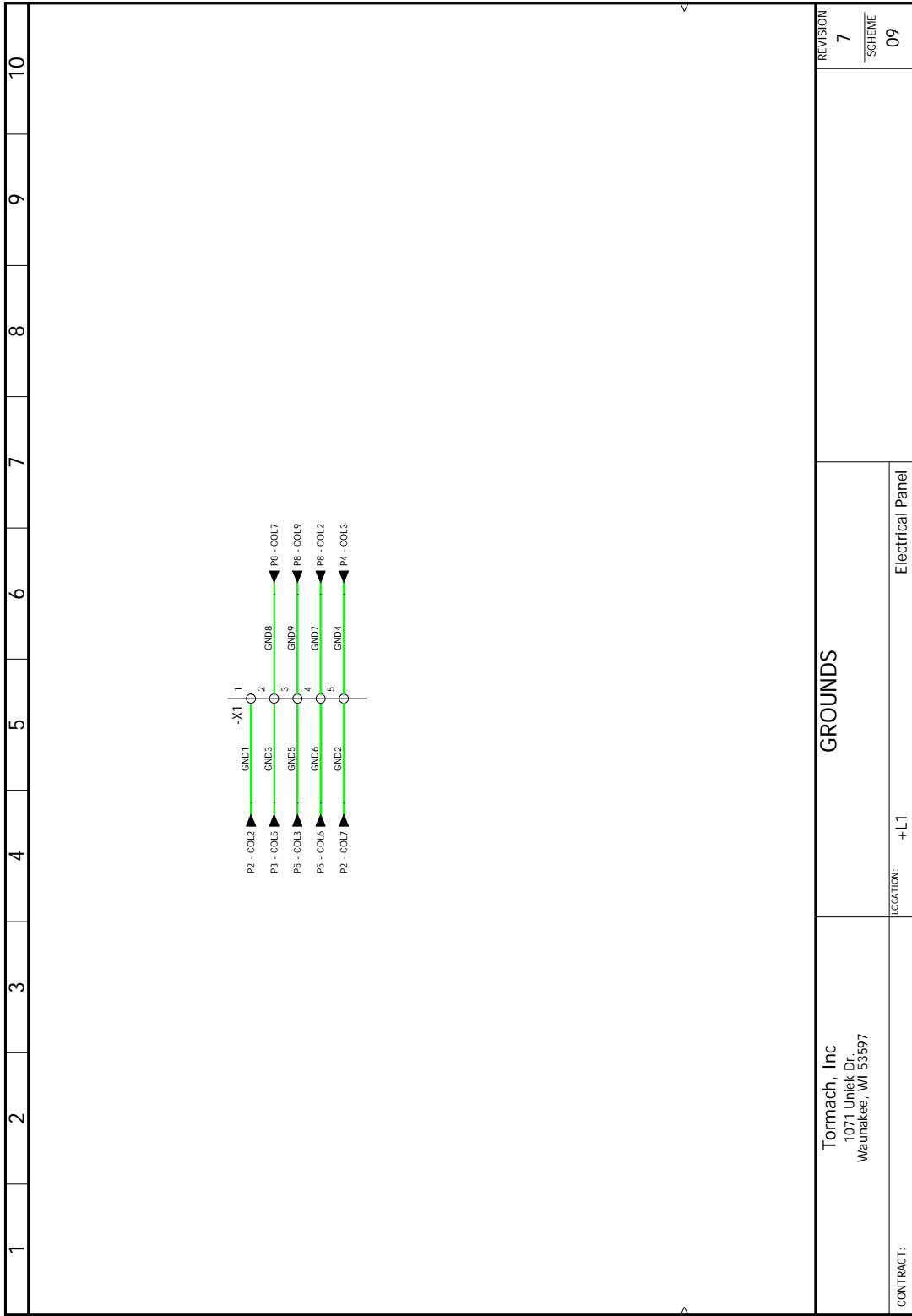
14.7 Accessory Power (Sheet 8)

14.7 ACCESSORY POWER (SHEET 8)



SOLIDWORKS Electrical

14.8 GROUNDS (SHEET 9)

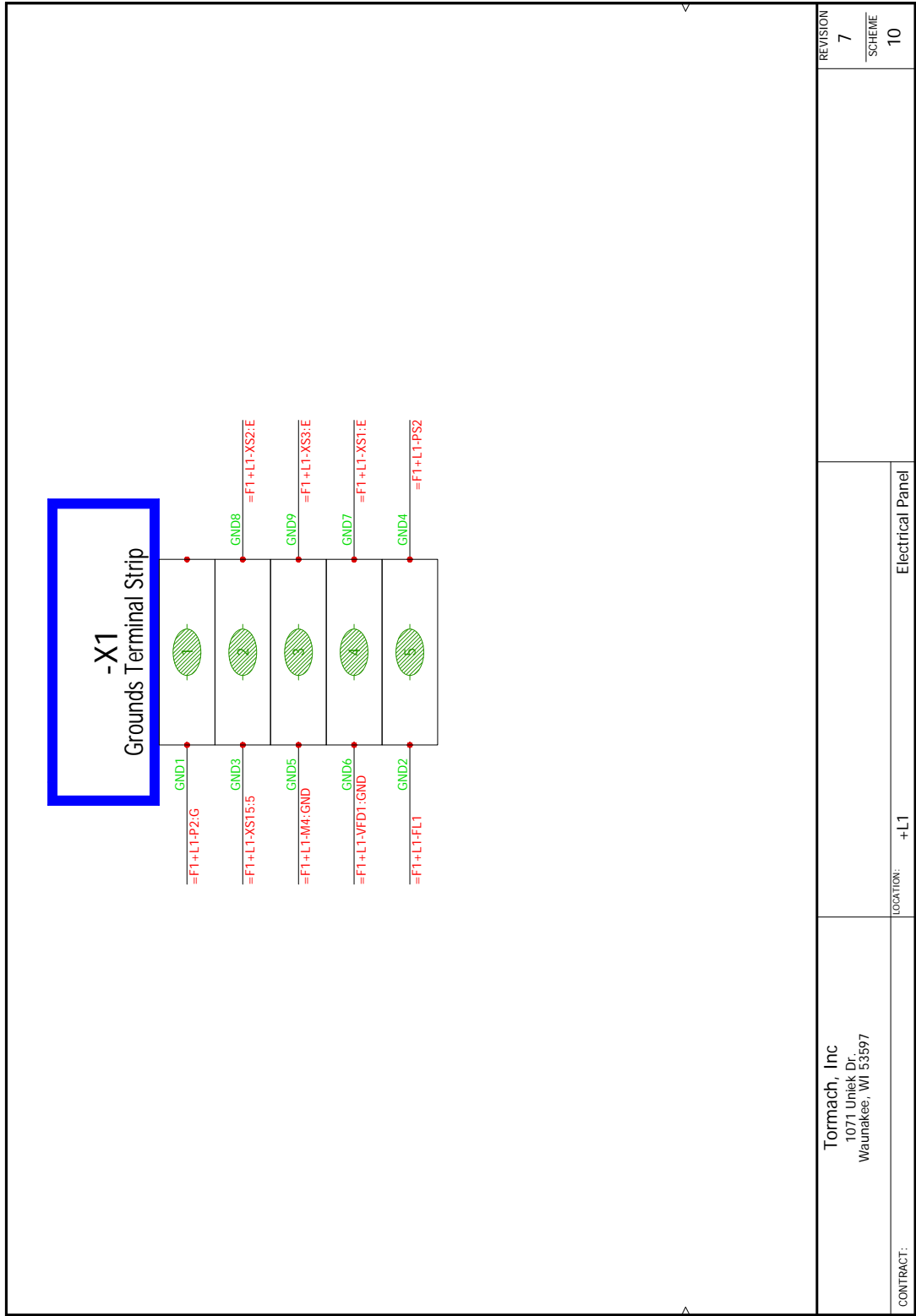


SOLIDWORKS Electrical

14: ELECTRICAL SCHEMATICS

14.9 Grounds Terminal Strip (Sheet 10)

14.9 GROUNDS TERMINAL STRIP (SHEET 10)

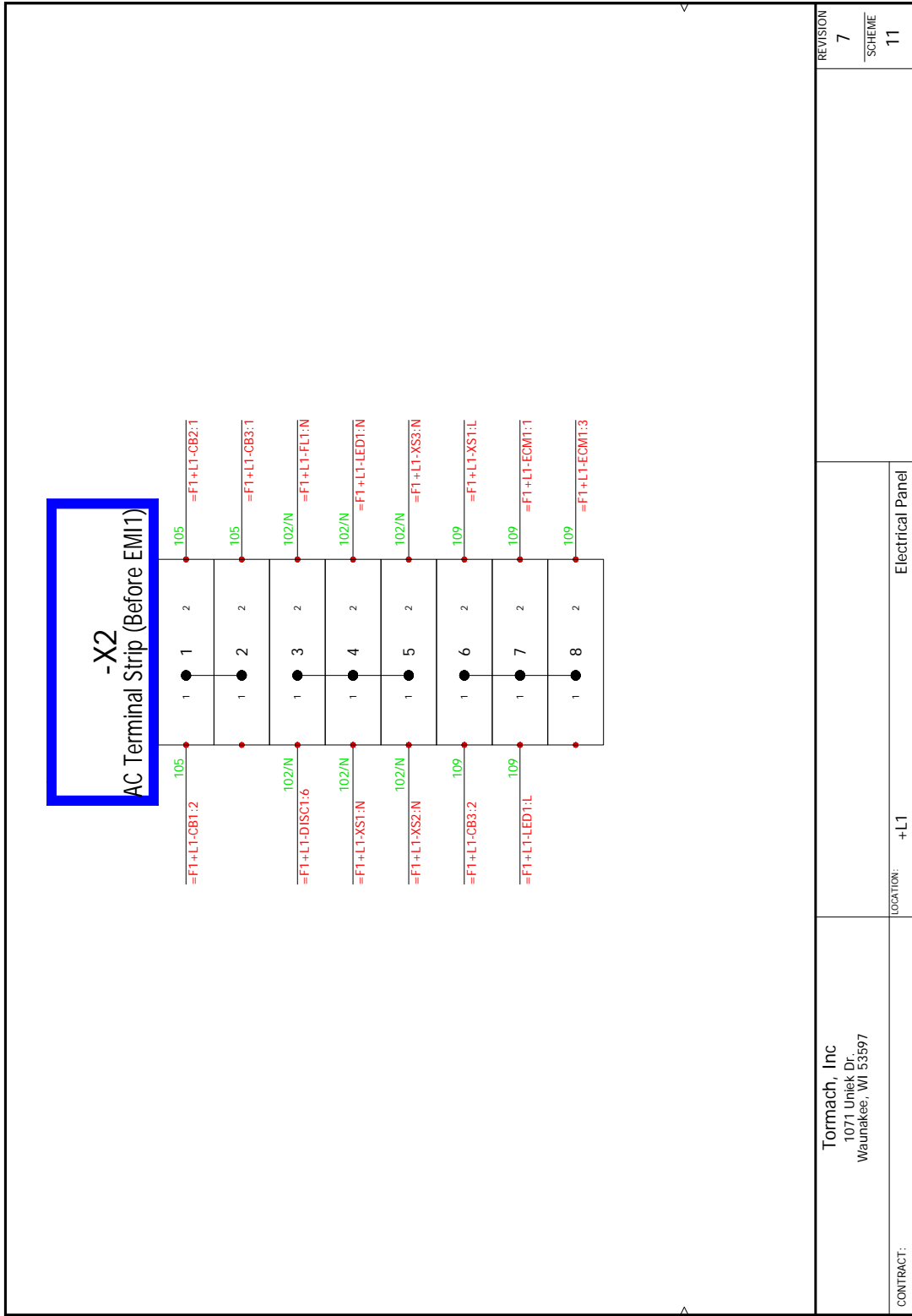


SOLIDWORKS Electrical

<p>Tormach, Inc 1071 Uniek Dr. Waunakee, WI 53597</p>	<p>REVISION 7</p>
	<p>SCHEME 10</p>
<p>CONTRACT:</p>	<p>LOCATION: +L1</p>
<p>Electrical Panel</p>	

Document created with version: 2025-0-0-5095-0

14.10 AC TERMINAL STRIP (BEFORE EMI1) (SHEET 11)



SOLIDWORKS Electrical

Tormach, Inc
 1071 Uniek Dr.
 Waunakee, WI 53597

LOCATION: +L1

Electrical Panel

REVISION
 7
 SCHEME
 11

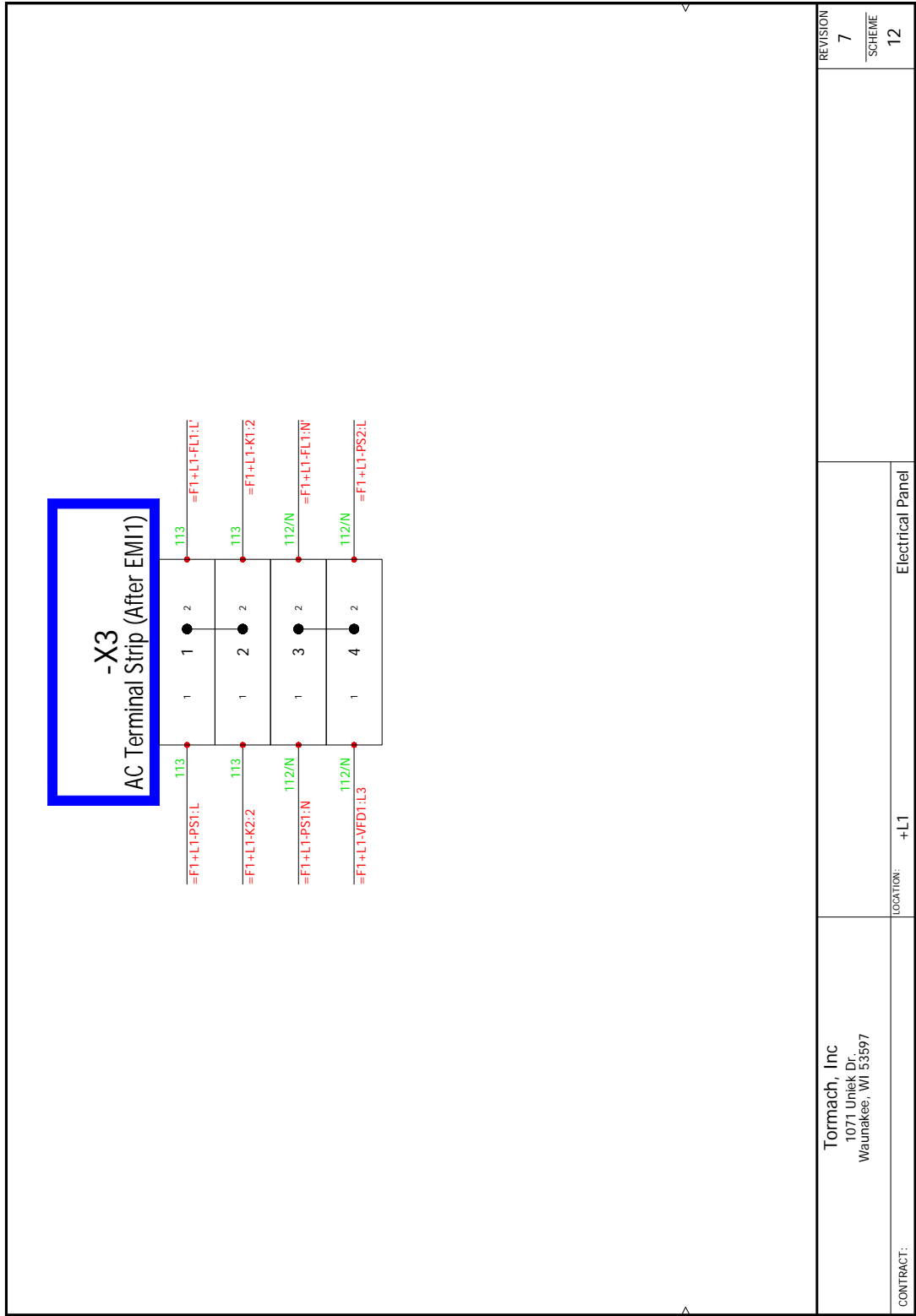
CONTRACT:

Document created with version: 2022.4.0.0.0000

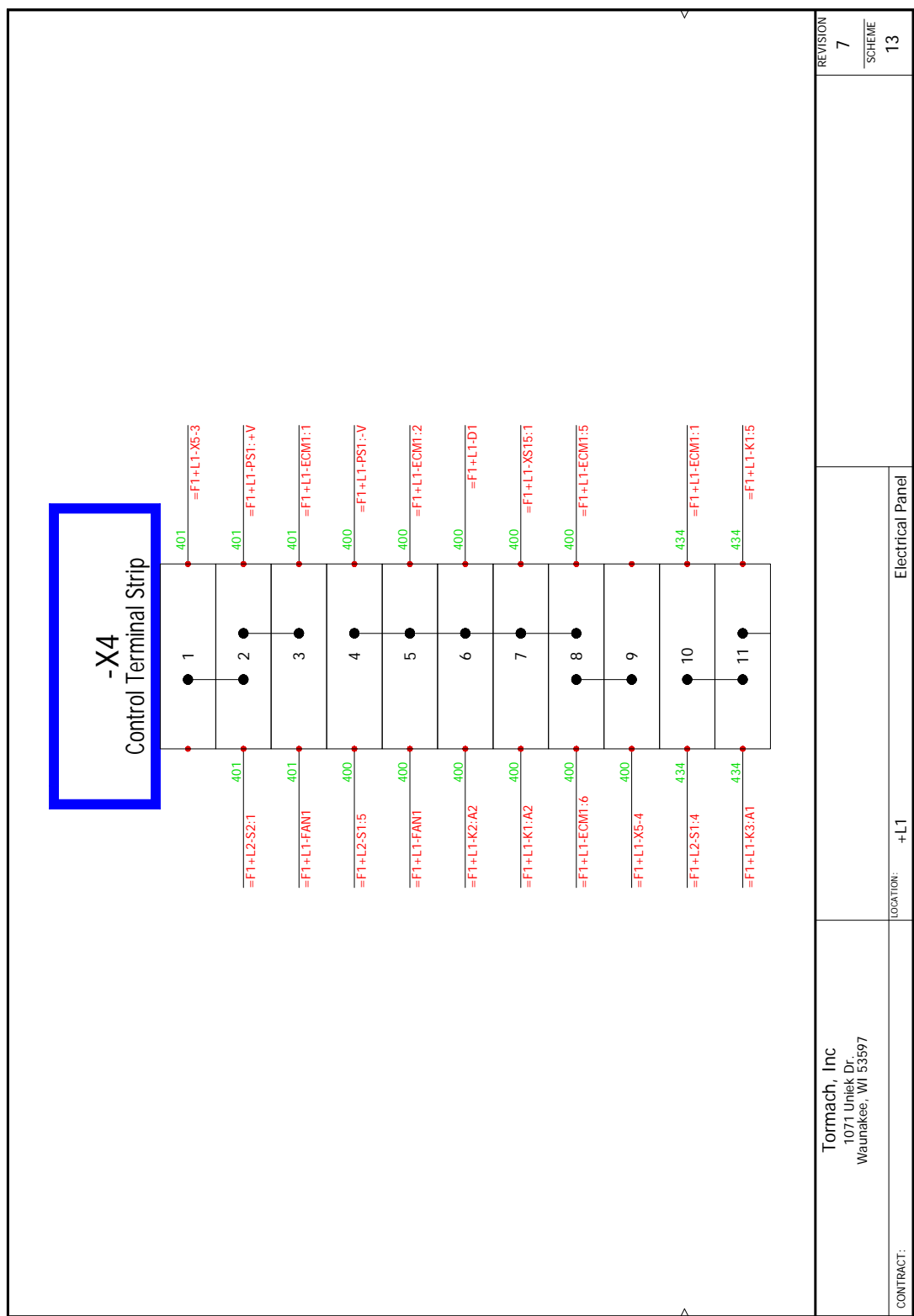
14: ELECTRICAL SCHEMATICS

14.11 AC Terminal Strip (After EMI1) (Sheet 12)

14.11 AC TERMINAL STRIP (AFTER EMI1) (SHEET 12)



14.12 CONTROL TERMINAL STRIP (SHEETS 13-15)

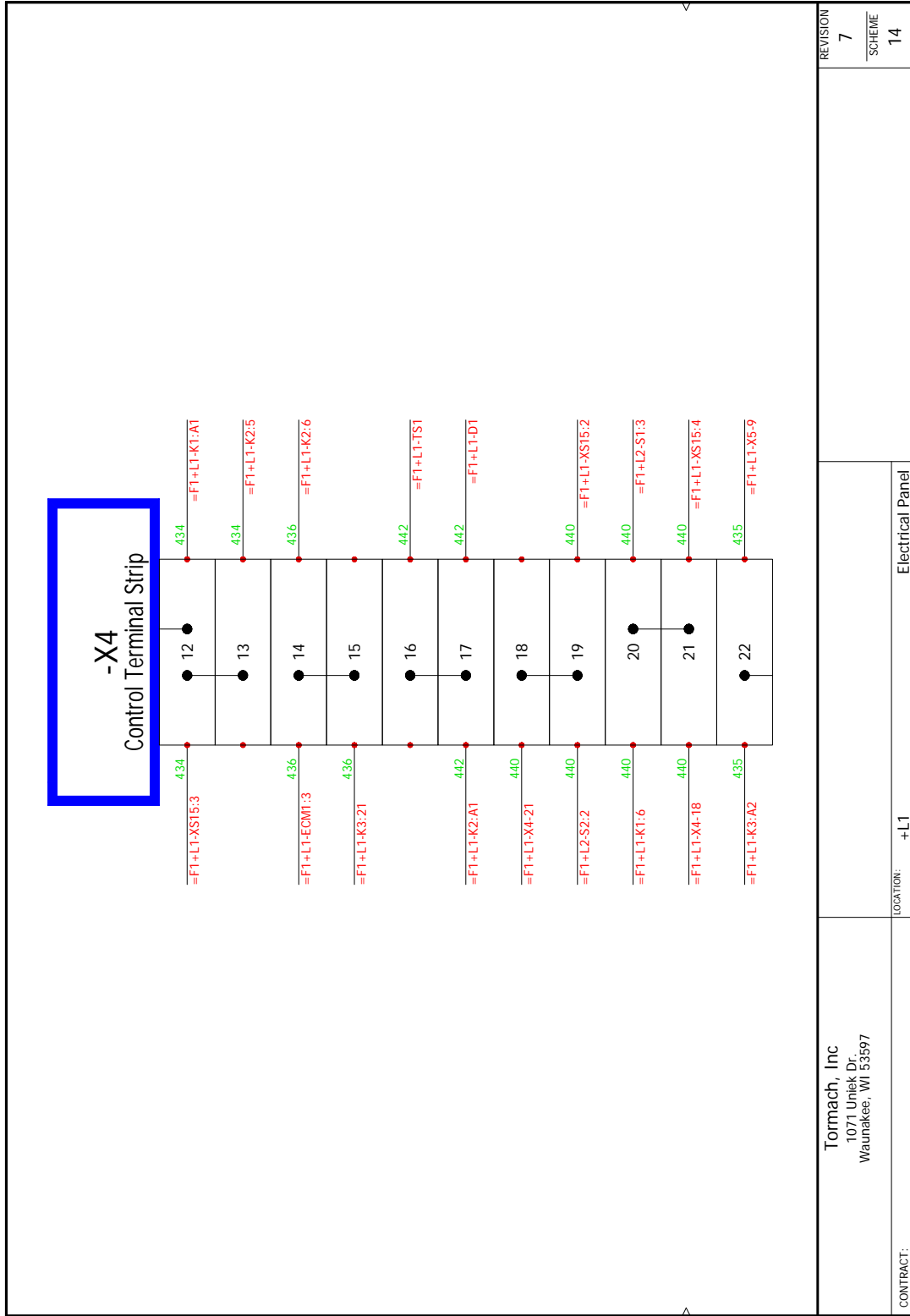


SOLIDWORKS Electrical

Tormach, Inc 1071 Uniek Dr. Waunakee, WI 53597	LOCATION: +L1	Electrical Panel	REVISION 7
	CONTRACT: <small>Document created with version: 2022.2.0.0.0000</small>		SCHEME 13

14: ELECTRICAL SCHEMATICS

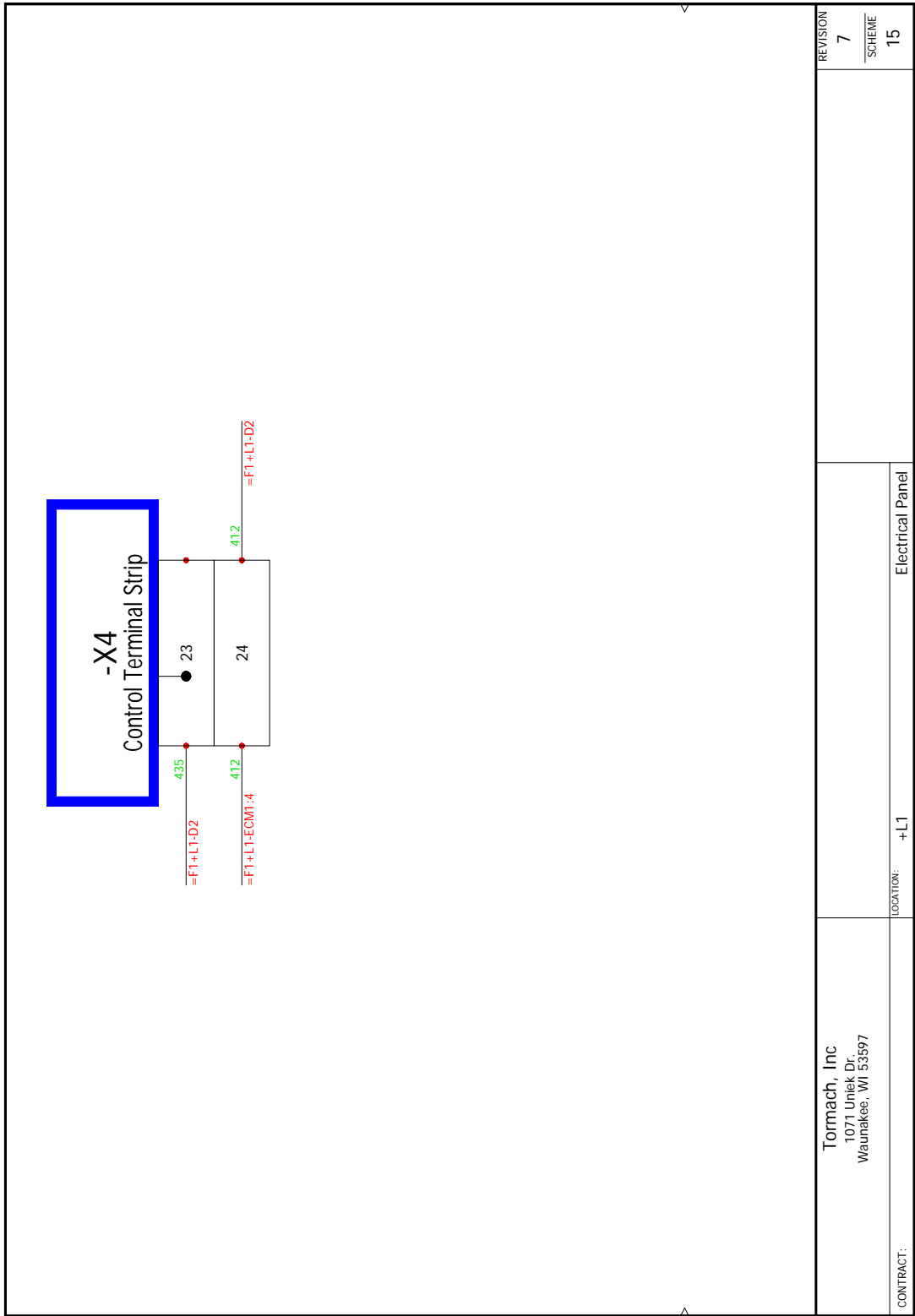
14.12 Control Terminal Strip (Sheets 13-15)



SOLIDWORKS Electrical

<p>CONTRACT: _____</p> <p>LOCATION: +L1</p> <p>Electrical Panel</p>		<p>REVISION</p> <p>7</p> <p>SCHEME</p> <p>14</p>
<p>Tormach, Inc 1071 Uniek Dr. Waunakee, WI 53597</p>		

Document created with version: 2025.0.0.5060



SOLIDWORKS Electrical

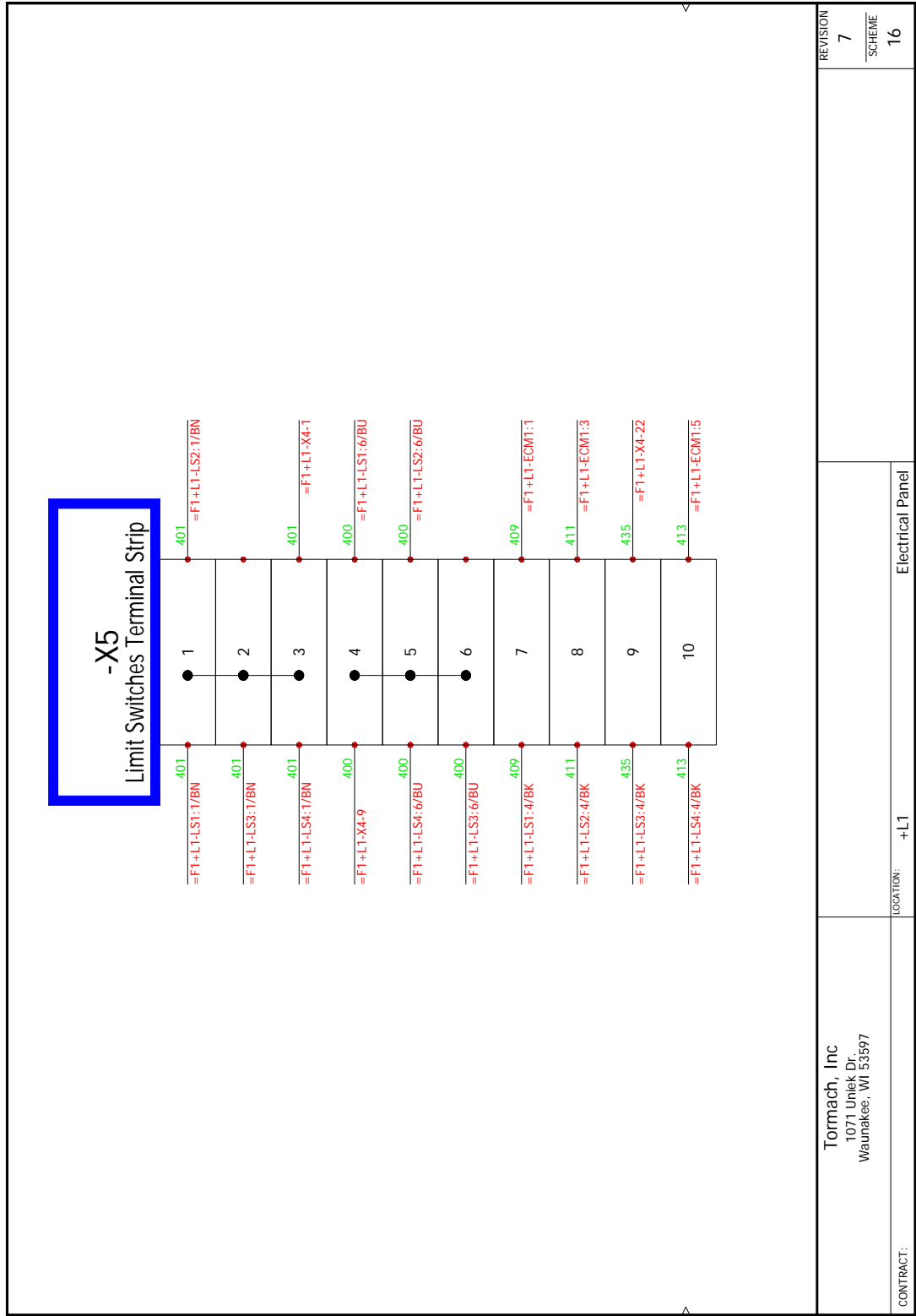
Tormach, Inc 1071 Uniek Dr. Waunakee, WI 53597	Electrical Panel	REVISION 7 SCHEME 15
CONTRACT:		LOCATION: +L1

Document created with version: 2022.4.0.0.0000

14: ELECTRICAL SCHEMATICS

14.13 Limit Switches Terminal Strip (Sheet 16)

14.13 LIMIT SWITCHES TERMINAL STRIP (SHEET 16)



<p>Tormach, Inc 1071 Uniek Dr. Waunakee, WI 53597</p>	<p>REVISION 7</p> <p>SCHEME 16</p>
<p>CONTRACT:</p>	<p>LOCATION: +L1</p> <p style="text-align: center;">Electrical Panel</p>

Document created with version: 2025-01-01-05050

SOLIDWORKS Electrical

