

TORMACH®
15Lx®



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®, 15LX®, 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.

DECLARATION OF CONFORMITY

This document contains a sample Declaration of Conformity. European users receive a signed and dated Declaration of Conformity with their machine.

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
2.5 Operating Modes Reference.....	26
2.5.1 Run Mode (Mode 1).....	26
2.5.2 Setup Mode (Mode 2).....	26
2.5.3 Mode Constraints.....	27
ABOUT YOUR MACHINE	29
3.1 Specifications.....	30
SITE REQUIREMENTS.....	31
4.1 General Site and Space Requirements.....	32
4.1.1 Site Requirements.....	32
4.1.2 Space Requirements.....	32
4.2 Electrical and Power Requirements.....	33
4.2.1 Electrical Requirements.....	33
4.2.2 Power Requirements.....	33
4.2.3 Options for Non-Conforming Sites.....	33

TABLE OF CONTENTS

4.3 Air Requirements	34
4.4 Space Planning Drawings	35
4.4.1 Machine Footprint	36
4.5 Electrical Site Plan Drawings	37
4.5.1 Electrical Site Plan	38
INSTALLATION	39
5.1 Before You Begin	40
5.2 Installation Tools and Items	41
5.3 Move the Pallet	42
5.4 Unpack the Machine Crate	43
5.5 Install Core Components	47
5.5.1 Assemble the Coolant Tank	47
5.5.2 Install the PathPilot Operator Console	47
5.5.3 Install the Air System Components	48
5.5.4 Set Up the Automatic Oiler	50
Specify the Interval Time	50
Specify the Actuation Time	51
5.6 Verify the Installation	52
5.6.1 Power on the Machine	52
5.6.2 Verify Axes Function	53
5.6.3 Verify Spindle Function	54
5.6.4 Power off the Machine	55
5.7 Install Accessory Components	56
5.7.1 Install the Automatic Collet Closer for 15LX	56
Required Tools	56
Before You Begin	56
Mount the Collet Closer Cylinder Assembly	56
Mount the Anti-Rotation Bracket and Solenoid	58
Make Air Connections	59
Make Electrical Connections	60
Align the Automatic Collet Closer Kit with the Spindle	60
Operation	61
Automatic Operation	61
Manual Operation	61
Switch Between ID and OD Clamping	61
Change the Collet	61
Adjust the Holding Force	62
Specifications	62
5.7.2 Connect the Collet Closer Foot Pedal Kit	62
5.8 Set Up the PathPilot Controller	63
5.8.1 Specify the Date and Time	63
5.8.2 Specify the Keyboard Language	63
5.8.3 Update PathPilot	63

SYSTEM BASICS	65
6.1 System Reference	66
6.1.1 Cross Slide Table	66
6.1.2 Spindle	66
<i>About the Spindle</i>	66
6.1.3 Axes	66
6.2 Tooling Reference	67
6.2.1 Tooling Styles	67
<i>Quick Change Tool Post Kit</i>	67
<i>8-position Turret</i>	67
<i>Gang Tooling</i>	67
6.2.2 Tool Post Locations	67
6.2.3 Tool Orientation	67
6.3 Basic Controls Reference	69
6.3.1 Machine Controls	69
6.3.2 PathPilot Interface	69
PATHPILOT INTERFACE OVERVIEW	71
7.1 About PathPilot	72
7.2 Notebook Section	73
7.2.1 Main Tab	73
7.2.2 File Tab	73
7.2.3 Settings Tab	74
7.2.4 Offsets Tab	74
7.2.5 Conversational Tab	74
7.2.6 Status Tab	74
7.3 Persistent Controls	76
7.3.1 Program Control Area	76
7.3.2 Position Status Area	76
7.3.3 Manual Control Area	77
7.4 Keyboard Shortcuts	78
7.5 Manage PathPilot Versions	79
7.5.1 Download and Install an Update File from the Controller	79
7.5.2 Install an Update File from a USB Drive	79
7.5.3 Install a Previous Version of an Update File	80
PATHPILOT TOOLS AND FEATURES	81
8.1 Create and Load G-Code Files	82
8.1.1 Load G-Code	82
<i>Transfer Files to and From the Controller</i>	82
<i>Preview G-Code Files</i>	82
<i>Access Recent G-Code Files</i>	83
<i>Close the Current Program</i>	83
8.1.2 Edit G-Code	83

TABLE OF CONTENTS

<i>Edit G-Code with a Text Editor</i>	83
<i>Edit G-Code with Conversational Programming</i>	83
Change the Step Order.....	84
Create a New Job Assignment.....	84
Load an Existing G-Code File.....	84
Edit a Job Assignment.....	84
8.1.3 Read G-Code.....	84
<i>Expand the G-Code Tab</i>	84
About the G-Code Tab.....	84
<i>Search in the Code</i>	85
<i>Set a New Start Line</i>	85
Lead-In Moves.....	86
<i>Change the View of the Tool Path Display</i>	86
About the Tool Path Display.....	87
8.1.4 Use Conversational Programming.....	87
<i>About Conversational Programming</i>	87
<i>Create an Outside Diameter</i>	87
About OD Turning.....	89
<i>OD Turning in PathPilot</i>	89
<i>Create an Internal Diameter</i>	89
Basic Internal Diameters.....	89
Extended Internal Diameters.....	90
About ID Turning.....	91
<i>ID Turning in PathPilot</i>	91
<i>Create a Profile on a Part</i>	92
Describe the Stock.....	92
Identify the Profile Points.....	93
Identify the Tool.....	93
Identify the Roughing and Finishing.....	93
Describe the Tool Geometry.....	94
About Profiling.....	94
<i>Create a Face on a Part</i>	94
About Facing.....	95
<i>Create a Chamfer or Radius on a Part</i>	96
About Chamfer and Radius.....	96
<i>Create a Groove or Part a Workpiece</i>	97
Create a Groove on a Part.....	98
Part a Workpiece from the Stock.....	98
About Grooving and Parting.....	98
<i>Create Holes on a Part or Tap a Hole</i>	99
Create a Hole on a Part.....	99
Create Threads in a Hole.....	99
About Drilling and Tapping.....	100
<i>Drilling</i>	100
<i>Tapping</i>	100

Create Threads on a Part	100
About Threading	101
8.2 Machine Settings and Accessories	103
8.2.1 Enable an Internet Connection	103
Enable Automatic Updates	103
8.2.2 Change the Network Name	103
8.2.3 Specify the Spindle Pulley Option	104
8.2.4 Specify the Tool Change Method	104
About Gang Tool Changes	104
About Manual Tool Changes	104
About Mixed Tooling Changes	104
8.2.5 Limit G30 Moves	104
About G30	105
8.2.6 Enable the On-Screen Keyboard	105
About Soft Keyboards	105
8.2.7 Enable Tooltips	105
8.2.8 Enable the USB M-Code I/O Interface Kit	106
8.2.9 Enable Feeds and Speeds Suggestions in Conversational Routines	106
8.2.10 Use a USB Camera	106
About USB Cameras	106
Manual Recording	107
Automatic E-Stop Loop Recording ("Dashcam")	107
Review Video and Image Files	108
File Naming Convention	108
G-Code Commands	108
File Naming Conventions	108
Use M01 to Take Pictures	108
8.2.11 Change the Screen Orientation	109
8.3 Set Up G-Code Programs	110
8.3.1 Set Tool Geometry Offsets	110
8.3.2 Touch Off the Tool Geometry Offsets	110
Touch X	110
Touch Z	111
About Tool Offsets	111
8.3.3 Set Work Offsets	111
About Work Offsets	111
8.3.4 View Work Offsets	112
8.3.5 View Available G-Code Modes	112
8.4 Run G-Code Programs	113
8.4.1 Bring the Machine Out of Reset	113
About Reset Mode	113
8.4.2 View the Active Axis to Jog	113
8.4.3 Jog the Machine	113
Jog in Continuous Velocity Mode	114
About Continuous Velocity Jogging	114

TABLE OF CONTENTS

- Jog in Step Mode.*114
 - About Step Jogging.*114
- 8.4.4 View the Current Machine Position 114
- 8.4.5 Reference the Machine.....115
 - About Referencing.*115
- 8.4.6 Start a Program.....115
 - About Cycle Start.*115
 - Cycle Start Reference*115
- 8.4.7 Stop Machine Motion 115
- 8.4.8 Operate the Coolant Pump.....115
 - About Coolant.*116
- 8.4.9 View the Active G-Code Modes116
- 8.4.10 View the Distance to Go116
 - About Distance to Go.*116
- 8.5 Control G-Code Programs.....117**
 - 8.5.1 Use the Feed Hold Function117
 - About Feed Hold.*117
 - 8.5.2 Use the Feed Rate Override Function.....117
 - About Feed Rate Override.*117
 - 8.5.3 Use M01 Break Mode.....118
 - About M01 Break.*118
 - 8.5.4 Use the Maxvel Override Function118
 - About Maxvel Override.*118
 - 8.5.5 Use Single Block Mode.....118
 - About Single Block.*118
 - 8.5.6 Use the Spindle Override Function118
 - About Spindle Override.*119
 - 8.5.7 Change the Feed Rate.....119
 - About Feed Rates.*119
 - 8.5.8 Change the Spindle Speed119
 - About Spindle Controls.*120
 - 8.5.9 Change the Feed Rate Mode.....120
 - About Feed Rate Modes.*120
 - Units per Minute Mode (G94).....121
 - Units per Revolution Mode (G95).....121
 - 8.5.10 Change the Spindle Speed Command Mode.....121
 - About Spindle Speed Command Modes.*121
 - Constant Surface Speed Mode (G96).....121
 - Constant RPM Mode (G97).....121
 - About Spindle Controls.*121
 - 8.5.11 Change the Tool Number.....122
 - 8.5.12 Use a G30 Position122
 - About G30.*123
 - 8.5.13 Manually Enter Commands.....123
 - About the MDI Line DRO Field.*123

<i>Admin Commands Reference</i>	123
8.5.14 Copy Recently Entered Commands.....	124
8.5.15 Use Feeds and Speeds Suggestions.....	124
<i>Refresh DRO Field Values</i>	125
<i>Enable Feeds and Speeds Suggestions in Conversational Routines</i>	125
<i>Create Tool Descriptions</i>	125
Manually Enter Tool Descriptions.....	125
Automatically Generate Tool Descriptions.....	126
<i>Tool Keywords Reference</i>	126
8.5.16 Use Cycle Counters (M30 and M99).....	127
<i>Monitor Cycle Counters</i>	128
<i>Change Cycle Counter Values</i>	128
8.6 System File Management	129
8.6.1 Manage System Files.....	129
8.6.2 Create Backup Files.....	129
<i>About Backup Files</i>	130
8.6.3 Restore Backup Files.....	130
8.6.4 Import and Export the Tool Table.....	131
<i>Import a .csv File</i>	131
<i>Export the Tool Table as a .csv File</i>	131
BASIC OPERATIONS	133
9.1 Start the Machine	134
9.2 Bring the Machine Out of Reset	135
9.2.1 About Reset Mode.....	135
9.3 Reference the Machine	136
9.3.1 About Referencing.....	136
9.4 Jog the Machine	137
9.4.1 About Jogging.....	137
9.4.2 About Continuous Velocity Jogging.....	137
9.4.3 About Step Jogging.....	137
9.4.4 Jog Controls Reference.....	138
<i>Jogging in PathPilot</i>	138
<i>Jogging with the Keyboard</i>	138
<i>Jogging with the (Optional) Jog Shuttle or Operator Console Pendant</i>	138
9.5 Manually Control the Spindle	139
9.5.1 About the Spindle.....	139
9.6 Load G-Code	140
9.6.1 Transfer Files to and From the Controller.....	140
9.7 Set Work Offsets	141
9.7.1 About Work Offsets.....	141
9.8 Set Tool Geometry Offsets	142
9.9 Touch Off the Tool Geometry Offsets	143
9.9.1 Touch X.....	143
9.9.2 Touch Z.....	143

TABLE OF CONTENTS

9.9.3 About Tool Offsets.....	143
9.9.4 Import and Export the Tool Table.....	144
<i>Import a .csv File.....</i>	<i>144</i>
<i>Export the Tool Table as a .csv File.....</i>	<i>144</i>
9.10 Operate the Coolant Pump.....	145
9.10.1 About Coolant.....	145
PROGRAMMING.....	147
10.1 Before You Begin.....	148
10.1.1 Definitions.....	148
<i>Linear Axes.....</i>	<i>148</i>
<i>Controlled Point.....</i>	<i>148</i>
<i>Coordinated Linear Motion.....</i>	<i>149</i>
<i>Feed Rate.....</i>	<i>149</i>
<i>Arc Motion.....</i>	<i>149</i>
<i>Dwell.....</i>	<i>149</i>
<i>Units.....</i>	<i>149</i>
<i>Current Position.....</i>	<i>149</i>
<i>Work Offsets.....</i>	<i>149</i>
<i>Selected Plane.....</i>	<i>149</i>
<i>Tool Table.....</i>	<i>149</i>
<i>Wear Offsets.....</i>	<i>149</i>
<i>Path Control Modes.....</i>	<i>150</i>
<i>Feed and Speed Override Controls.....</i>	<i>150</i>
<i>Block Delete Control.....</i>	<i>150</i>
<i>Optional Program Stop Control.....</i>	<i>150</i>
10.2 Programming Overview.....	151
10.2.1 About G-Code Programming Language.....	151
10.2.2 G-Code Formatting Reference.....	151
<i>Line Numbers.....</i>	<i>151</i>
<i>Words.....</i>	<i>151</i>
<i>Letters.....</i>	<i>152</i>
<i>Values.....</i>	<i>152</i>
<i>Order of Execution.....</i>	<i>153</i>
<i>Modal Groups.....</i>	<i>154</i>
<i>Comments.....</i>	<i>155</i>
10.2.3 Supported G-Codes Reference.....	155
10.3 Programming G-Code.....	156
10.3.1 About the Examples Used.....	156
10.3.2 Rapid Linear Motion (G00).....	156
10.3.3 Linear Motion at Feed Rate (G01).....	156
10.3.4 Arc at Feed Rate (G02 and G03).....	157
<i>Radius Format Arc.....</i>	<i>157</i>
<i>Center Format Arc.....</i>	<i>157</i>
10.3.5 Dwell (G04).....	157

10.3.6 Lathe Diameter Mode (G07).....	157
10.3.7 Set Offsets (G10).....	157
<i>Set Tool Table (G10 L1)</i>	158
<i>Set Coordinate System (G10 L2)</i>	158
<i>Set Tool Table (G10 L10)</i>	158
<i>Set Tool Table (G10 L11)</i>	158
<i>Set Coordinate System (G10 L20)</i>	159
10.3.8 Plane Selection (G17, G18, G19).....	159
10.3.9 Length Units (G20 and G21).....	159
10.3.10 Return to Predefined Position (G28 and G28.1).....	159
10.3.11 Return to Predefined Position (G30 and G30.1).....	159
10.3.12 Spindle-Synchronized Motion (G33).....	160
10.3.13 Rigid Tapping (G33.1).....	160
10.3.14 Cutter Compensation (G40, G41, G42).....	161
10.3.15 Dynamic Cutter Compensation (G41.1 and G42.1).....	161
10.3.16 Absolute Coordinates (G53).....	161
10.3.17 Select Work Offset Coordinate System (G54 to G54.1 P500).....	161
10.3.18 Set Exact Path Control Mode (G61).....	162
10.3.19 Set Blended Path Control Mode (G64).....	162
10.3.20 High Speed Peck Drill (G73).....	162
10.3.21 Threading Cycle (G76).....	162
<i>Optional Settings</i>	163
10.3.22 Tapping Cycle (G84).....	163
10.3.23 Distance Mode (G90 and G91).....	164
10.3.24 Arc Distance Mode (G90.1 and G91.1).....	164
10.3.25 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	164
10.3.26 Feed Rate Mode (G93, G94, and G95).....	165
10.3.27 Spindle Control Mode (G96 and G97).....	165
10.4 Programming M-Code	166
10.4.1 Supported M-Codes Reference.....	166
10.4.2 Program Stop and Program End (M00, M01, M02, and M30).....	166
10.4.3 Display Information and Capture Images During an M00 or M01 Break.....	167
<i>Display Information with Images</i>	167
<i>Display Information with Text</i>	167
<i>Capture Images with a USB Camera</i>	167
10.4.4 Spindle Control (M03, M04, and M05).....	167
10.4.5 Coolant Control (M08 and M09).....	167
10.4.6 Automatic Collet Closer Control (M10 and M11).....	167
10.4.7 Override Control (M48 and M49).....	168
10.4.8 Feed Override Control (M50).....	168
10.4.9 Spindle Speed Override Control (M51).....	168
10.4.10 Set Current Tool Number (M61).....	168
10.4.11 Set Output State (M64 and M65).....	168
10.4.12 Call Subroutine (M98).....	168
10.4.13 Return from Subroutine (M99).....	169

TABLE OF CONTENTS

10.4.14 USB Camera Control (M301, M302, M303).....	169
10.5 Programming Input Codes.....	170
10.5.1 Feed Rate (F).....	170
10.5.2 Spindle Speed (S).....	170
10.5.3 Change Tool Number (T).....	170
10.6 Advanced Programming.....	171
10.6.1 Parameters.....	171
<i>Parameters Reference.....</i>	<i>171</i>
Parameter Syntax.....	171
Parameter Scope.....	171
Behavior of Uninitialized Parameters.....	171
Parameter Mode.....	171
Persistence and Volatility.....	171
Intended Use.....	171
<i>Numbered Parameters Reference.....</i>	<i>172</i>
Read-Only Parameters.....	172
<i>Subroutine Parameters Reference.....</i>	<i>172</i>
<i>Named Parameters Reference.....</i>	<i>172</i>
10.6.2 Expressions.....	173
<i>Binary Operators Reference.....</i>	<i>173</i>
<i>Functions Reference.....</i>	<i>173</i>
10.6.3 Subroutines.....	174
<i>Subroutines Reference.....</i>	<i>174</i>
<i>Conditional Subroutines Reference.....</i>	<i>175</i>
if/endif.....	175
if/elseif/else/endif.....	175
<i>Repeating Subroutines Reference.....</i>	<i>175</i>
<i>Looping Subroutines Reference.....</i>	<i>175</i>
do/while.....	176
while/endwhile.....	176
MACHINE MAINTENANCE.....	177
11.1 Maintenance Safety.....	178
11.1.1 All Maintenance Procedures.....	178
11.1.2 Swarf Maintenance Procedures.....	178
11.1.3 Coolant Maintenance Procedures.....	178
11.2 Maintenance Schedules.....	179
11.2.1 Daily.....	179
11.2.2 Weekly.....	179
11.2.3 Monthly.....	179
11.2.4 Semi-Annually.....	179
11.2.5 Annually.....	179
11.3 Clean the Z-Axis Wipers.....	180
11.4 Clean the Coolant System.....	181
11.4.1 About Cutting Fluid.....	181

11.4.2 Cutting Fluid Reference.....	181
11.5 Examine the Enclosure Windows.....	182
11.6 Examine the Spindle Belt.....	183
11.7 Lubricate the Machine.....	184
11.7.1 About the Automatic Oiler.....	184
11.7.2 Lubrication System Reference.....	184
11.8 Prevent Rust.....	185
11.8.1 Remove Rust.....	185
11.9 Adjust the Gibs.....	186
TROUBLESHOOTING.....	187
12.1 Troubleshooting Safety.....	188
12.2 Getting Help.....	189
12.3 Required Tools.....	190
12.4 Frequently Found Problems.....	191
12.5 Electrical Service.....	192
12.5.1 LED Identification.....	192
12.5.2 Overview.....	193
12.6 Power Distribution Subsystem.....	194
12.6.1 The Machine Won't Power On.....	194
12.6.2 The PathPilot Operator Console Won't Power On.....	194
12.7 Control Power Subsystem.....	196
12.7.1 The Machine Won't Power On.....	196
12.8 Servo Drive Subsystem.....	197
12.8.1 All Axes Won't Move When Commanded.....	197
12.8.2 One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move.....	198
12.8.3 Axis Movement is Noisy.....	199
12.8.4 Lost Motion on Axis Travel.....	200
12.9 Spindle Drive Subsystem.....	202
12.9.1 The Spindle Won't Turn.....	202
12.9.2 Machining Operations are Loud ("Chatter").....	204
12.10 Servo IO Breakout Board Subsystem.....	205
12.10.1 Groups of Inputs and Outputs Aren't Functioning.....	205
<i>Servo Drive Inputs and Outputs.....</i>	<i>205</i>
DIAGRAMS AND PARTS LISTS.....	207
13.1 Base Machine Assembly.....	208
13.2 Machine Base (with Motors).....	210
13.3 Bed Casting Assembly.....	212
13.4 Z-Axis Saddle Assembly.....	214
13.5 X-Axis Cross Slide Assembly.....	216
13.6 Headstock Assembly.....	218
13.7 Stand Assembly.....	221
13.7.1 Electrical Cabinet Door Assembly.....	223
13.8 Electrical Cabinet Assembly.....	225

TABLE OF CONTENTS

13.8.1 Exterior Layout	225
13.8.2 Interior Layout	227
13.9 Machine Pneumatics Distribution Assembly.....	230
13.10 Enclosure Assembly.....	232
13.10.1 Pivot Arm for Console Assembly.....	233
13.10.2 Enclosure Left Side Assembly.....	235
13.10.3 Enclosure Right Side Assembly.....	237
13.10.4 Enclosure Door Assembly.....	240
13.11 Flood Coolant Kit Assembly.....	242
<i>Flood Coolant Filter Assembly.....</i>	<i>244</i>
<i>Coolant Tank Assembly.....</i>	<i>246</i>
13.12 Accessory Diagrams and Parts Lists.....	248
13.12.1 Pneumatics Diagram.....	249
13.12.2 Automatic Collet Closer Assembly.....	251
13.12.3 Air Gun Kit Assembly.....	253
ELECTRICAL SCHEMATICS.....	255
14.1 230 Vac Power (Sheet 1).....	256
14.2 24 Vdc Power (Sheet 2).....	257
14.3 Safety Systems (Sheet 3).....	258
14.4 Spindle Drive (Sheet 4).....	259
14.5 XZ Axis Drives (Sheet 5).....	260
14.6 Turret Drive (Sheet 6).....	261
14.7 IO Board – Power, Safety, Aux Outputs (Sheet 7).....	262
14.8 IO Board – Digital Inputs and Outputs (Sheet 8).....	263
14.9 IO Board – Aux Inputs, Brakes, Coolant (Sheet 9).....	264
14.10 Accessory / Aux (Sheet 10).....	265
14.11 Grounds (Sheet 11).....	266
14.12 AC Terminal Strip (-X1) (Sheets 12-14).....	267
14.13 Control Terminal Strip (-X2) (Sheet 15).....	270
14.14 Grounds Terminal Strip (-X3) (Sheet 16).....	271
14.15 Bill of Materials (Sheet 17-18).....	272

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
2.5 Operating Modes Reference	26

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! Electrocution 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

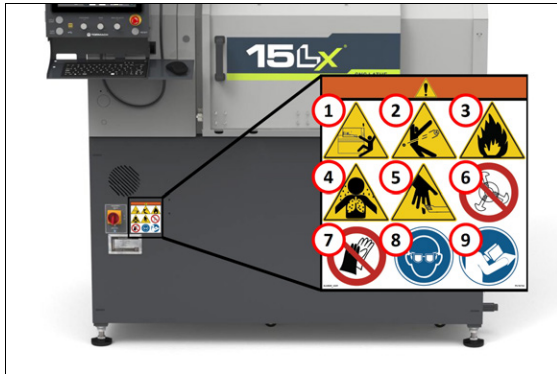


Figure 2-2: Example of a safety decal next to the Main Disconnect.

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 Z87+.
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

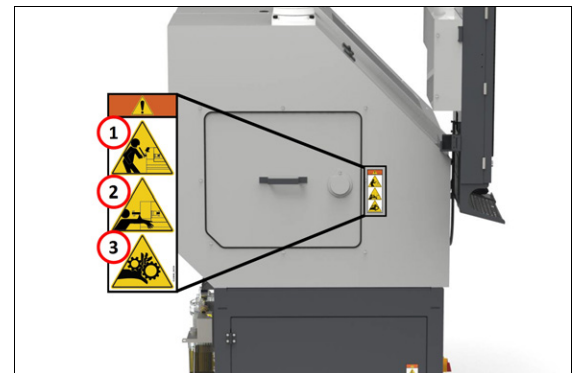


Figure 2-3: Example of safety decals on the rear of the headstock.

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: SAFETY

2.3 Safety Overview

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.
3. **WARNING! Moving Parts Hazard.** Moving parts can entangle, pinch, or cut you, causing death or serious injury. Before operating this machine in any way, you must verify that all body parts, long hair, and clothes are clear of the machine's extent of motion.

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 or the chuck guard 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 33).

- ✓ Confirm that the machine installation meets all codes and regulations of your locality.
- ✓ Confirm that electrical connections are performed by a certified electrician.
- ✓ Lock the electrical cabinet door and remove the keys when the machine is not being serviced to prevent unqualified or unauthorized personnel from accessing the electrical cabinet.
- ✗ Never operate the machine with the electrical cabinet door open.
- ✗ Never reach into the electrical cabinet with the machine powered on.
- ✗ Never modify the machine's electronics.
- ✗ Never drill into the electrical cabinet.

2: SAFETY

2.5 Operating Modes Reference

2.5 OPERATING MODES REFERENCE

This machine has two modes: run mode and setup mode. In both modes, you must close and lock the enclosure doors before starting a program. In setup mode, experienced operators can perform maintenance or set up jobs while the enclosure doors are open. To access setup mode, you must have a key.

For unrestricted machine motion to occur, you must do the following in the order listed:

- 1. Close the enclosure doors.
- 2. Command the machine to move or start a program. The enclosure doors automatically lock before any machine motion occurs.

2.5.1 Run Mode (Mode 1)

When the enclosure doors are open, and the machine is in run mode, no motion can occur. For example, you can't jog the axis or control the spindle.

2.5.2 Setup Mode (Mode 2)

It's the machine owner's responsibility to verify that only personnel who are trained in operation, maintenance, or both shall access setup mode. Setup mode is intended only for qualified machinery operators and maintenance professionals, and only to set up a program or do maintenance procedures. When you're not using setup mode, you must verify that the machine is in run mode.

- **Mode Identification** To identify which mode the machine is in, read the **Status** read-only field.



Figure 2-5: PathPilot indicates the controller's current mode.

- **Key Use** To access setup mode, you must have a key. Use the key on the top of the operator console to switch between modes: turn the key to **0** for run mode, and **I** for

setup mode.

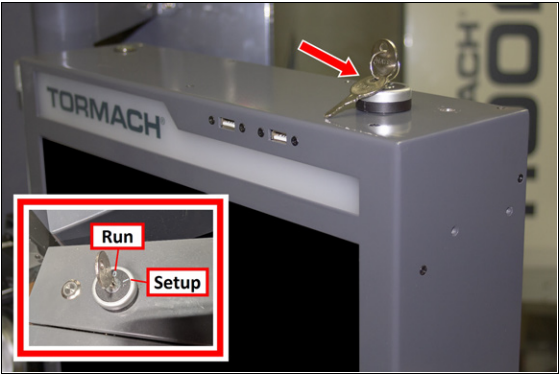


Figure 2-6: Setup mode key in the operator console.

Note: While in setup mode, you can't remove the key. Turn the key to run mode to remove the key from the operator console.

- **Hold-to-Run Device Use** For machine motion to occur in setup mode, you must use a hold-to-run device. Hold-to-run devices include:
 - The button on the side of the operator console's pendant.

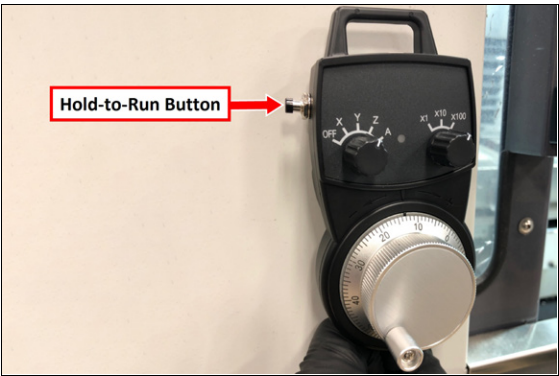


Figure 2-7: Hold-to-run button on the pendant.

- The jog shuttle wheel (the outer portion), which springs back to neutral when you release it.

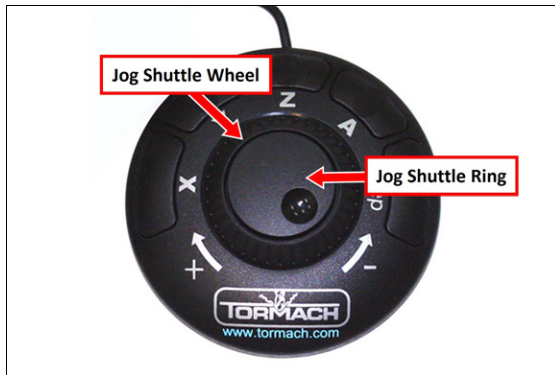


Figure 2-8: Jog shuttle components.

- **Machine Motion Restrictions** When the enclosure doors are open, and the machine is in setup mode, machine motion is restricted:
 - **Maximum Feed Rate** 2 m/min, on a maximum of one axis, with a hold-to-run device.



Note: To jog an axis with the keyboard, also use a hold-to-run device.

- **Maximum Spindle Speed** 700 rpm with a hold-to-run device.

2.5.3 Mode Constraints

Both operating modes restrict some machine activities while either the spindle door or the enclosure doors are open. To understand how to use the machine with the doors open, read the following tables.



Note: This table assumes that the door is open and stays open during each activity.

To ... with the Enclosure Doors Open	In Run Mode	In Setup Mode
Rotate the turret	✗	✗
Open or close the automatic collet closer	✓	✓
Start a program	✗	✗

To ... with the Enclosure Doors Open	In Run Mode	In Setup Mode
Issue an MDI command	✗	✗
Use coolant	✓	✓
Use a hold-to-run device to move one axis	✗	✓
Use a hold-to-run device to move more than one axis	✗	✗
Use a hold-to-run device to rotate the spindle less than 700 rpm	✗	✓
Use a hold-to-run device to rotate the spindle more than 700 rpm	✗	✗
To ... with the Spindle Doors Open	In Run Mode	In Setup Mode
Rotate the turret	✓	✓
Open or close the automatic collet closer	✓	✓
Start a program	✗	✗
Issue an MDI command	✗	✗
Use coolant	✓	✓
Use a hold-to-run device to move one axis	✗	✓
Use a hold-to-run device to move more than one axis	✗	✗
Use a hold-to-run device to rotate the spindle less than 700 rpm	✗	✗
Use a hold-to-run device to rotate the spindle more than 700 rpm	✗	✗

Key to the symbols used in the above table:

Symbol	Meaning of Symbol
✓	Allowed

2: SAFETY

2.5 Operating Modes Reference

Symbol	Meaning of Symbol
	Not allowed

ABOUT YOUR MACHINE

IN THIS SECTION, YOU'LL LEARN:

- About this machine's specifications.

CONTENTS

3.1 Specifications	30
--------------------------	----

3: ABOUT YOUR MACHINE

3.1 Specifications

3.1 SPECIFICATIONS

Travels	
X-Axis	10" (254 mm)
Z-Axis	14" (356 mm)
Spindle	
Continuous Spindle Power	6 hp (4.5 kW)
Peak Spindle Power	10 hp (7.4 kW)
Maximum Spindle Speed	3500 rpm
Spindle Taper	5C (with removable insert)
Spindle Bore Diameter	1.53" (38.86 mm) (with 5C insert removed)
Spindle Nose	D1-4 camlock mount
Motion	
Maximum Feed Rate (X- and Z-Axis)	360 IPM (9.14 m/min)
EtherCAT Motion System	Closed-loop, true position, high-performance servos
Machine Specifications	
Machine Footprint	63" × 47" (1.6 m × 1.2 m)
Overall System Height	73" (1.9 m)
Typical System Weight	With gang tooling: 1631 lb (740 kg) With a turret: 1764 lb (800 kg)
Swing Over Bed	15" (381 mm)
Swing Over Carriage	6.2" (157.5 mm)
Recommended Chuck Size	6" (152 mm)
Coolant Tank Capacity	19 gal (71 liters)
Power	
Primary Power Required	Single-Phase 230 Vac, 50/60 Hz
Recommended Circuit Amperage	30 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	32
4.2 Electrical and Power Requirements	33
4.3 Air Requirements	34
4.4 Space Planning Drawings	35
4.5 Electrical Site Plan Drawings	37



TORMACH®

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 230 Vac, 50/60 Hz
 - **Recommended Circuit Amperage** 30 A breaker



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

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

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** 63" × 47" (1.6 m × 1.2 m)
- **Machine Height** 73" (1.9 m)

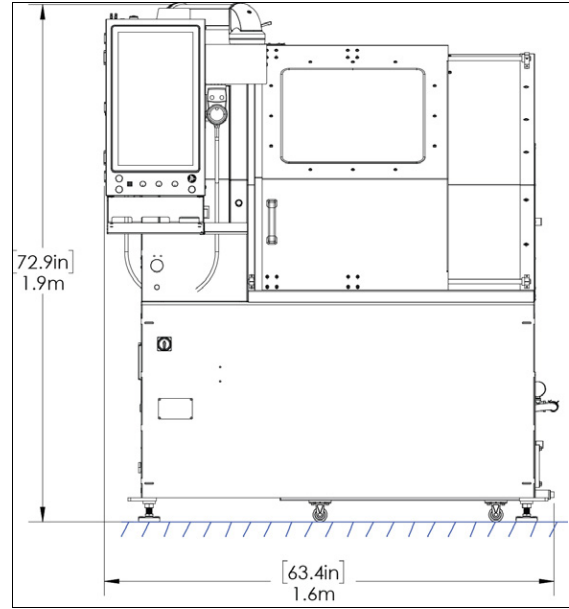


Figure 4-1: Dimensions of the machine itself, as viewed from the front.

- **Typical System Footprint** 130" × 63" (3.3 m × 1.6 m)

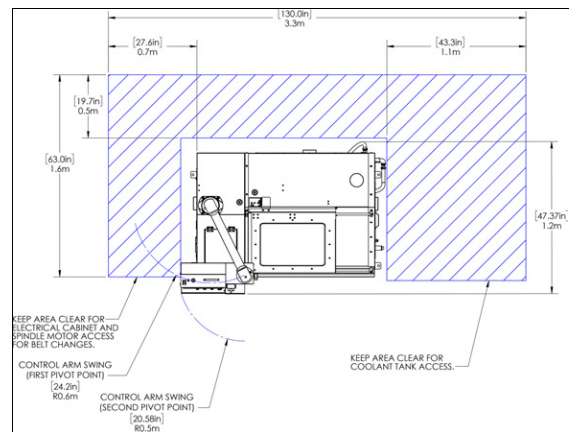


Figure 4-2: Dimensions of the machine and its required added space, as viewed from above.

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 230 Vac, 50/60 Hz
- **Recommended Circuit Amperage** 30 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 L6-30P plug, designed for use with a NEMA L6-30R 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.

- **Buck-Boost Transformer** Used to adjust line voltages. While the machine can run on line voltages between 200 to 240 Vac, spindle performance is reduced on line voltages below 230 Vac, and damage to electrical

components is possible on line voltages above 240 Vac. To prevent reduction in spindle performance or damage to electrical components, we recommend the Buck-Boost Transformer (PN 32554).

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.
Use the pre-installed FRL Filter-Regulator-Lubricator for this purpose.

4.4 SPACE PLANNING DRAWINGS

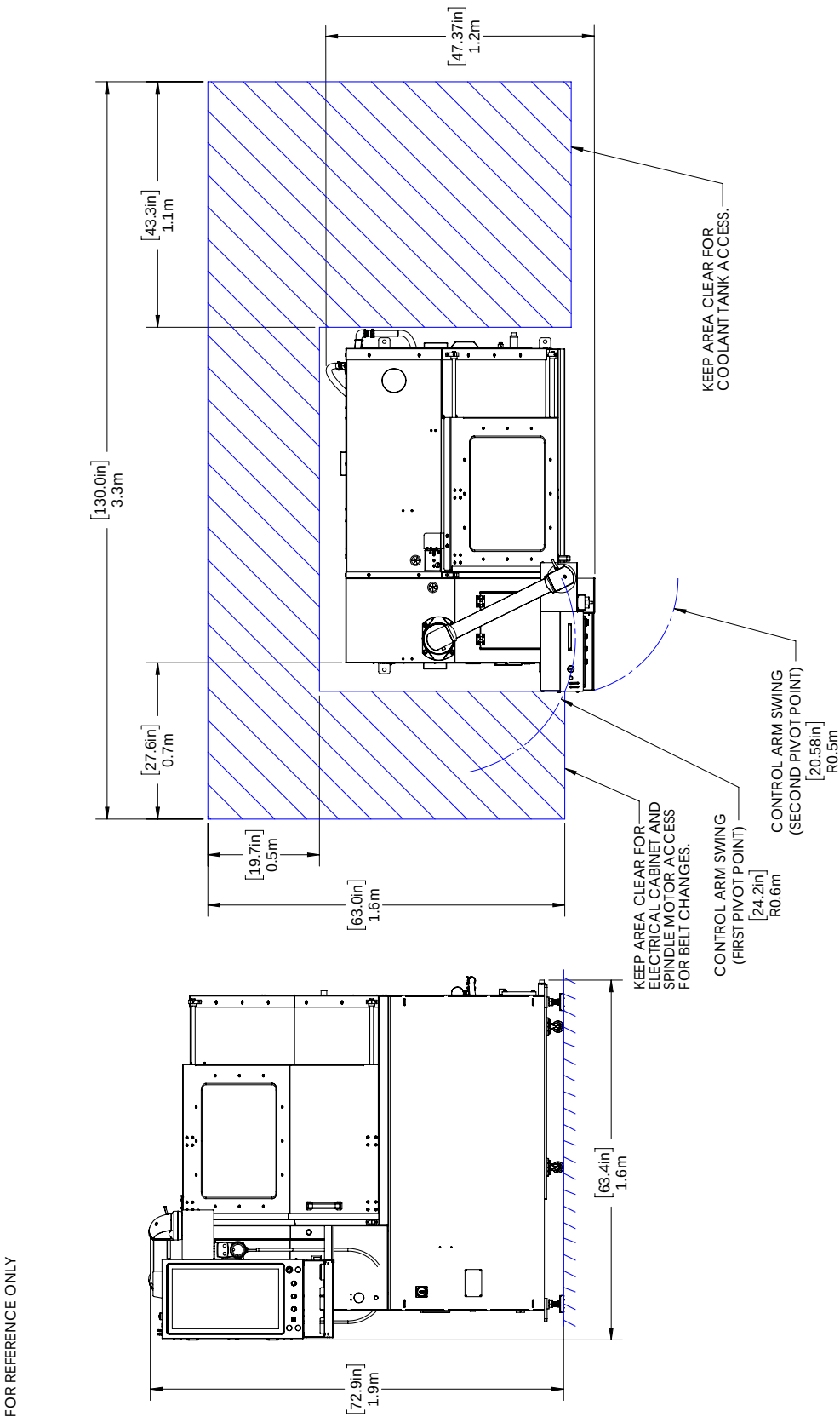
|

4.4.1 Machine Footprint 36

4: SITE REQUIREMENTS

4.4 Space Planning Drawings

4.4.1 Machine Footprint



4.5 ELECTRICAL SITE PLAN DRAWINGS

4.5.1 Electrical Site Plan	38
---	-----------

4: SITE REQUIREMENTS

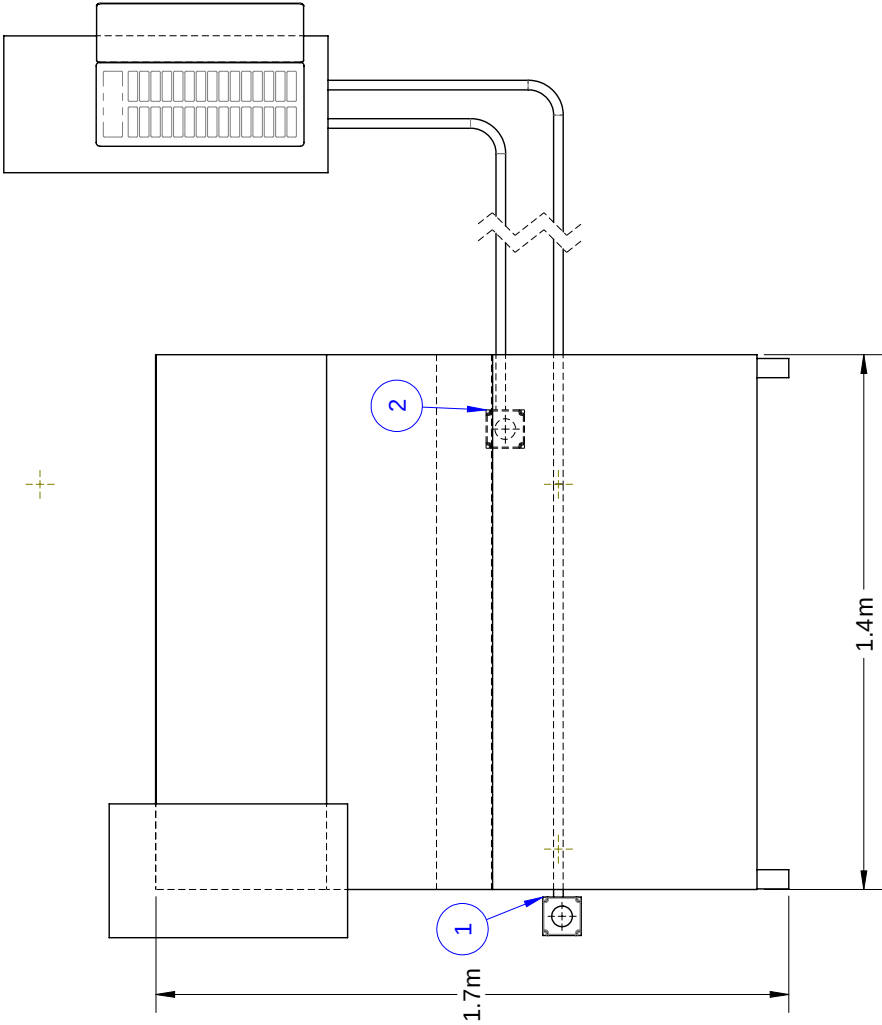
4.5 Electrical Site Plan Drawings

4.5.1 Electrical Site Plan

NOTES

1. ELECTRICAL INSTALLATION MUST BE COMPLETED BY A QUALIFIED ELECTRICIAN IN ACCORDANCE WITH ALL APPLICABLE CODES AND REGULATIONS
2. CONFIRM THAT THE MAINS VOLTAGE CONFORMS TO REQUIREMENTS BEFORE CONNECTING THE MACHINE OR ACCESSORIES
3. NOMINAL VOLTAGE. VOLTAGES AS LOW AS 208V ARE PERMISSIBLE BUT WILL REDUCE SPINDLE PERFORMANCE.

ITEM NO.	DEVICE	VOLTAGE	RUNNING WATTAGE	PEAK WATTAGE	BREAKER	CONNECTOR
1	BASE MACHINE • LATHE • CONTROLLER • LIGHTING • COOLANT	240V (1Ø) 	5.4kW	12.0kW	30A	NEMA L6-30P
2	OIL SKIMMER (OPTION)	120V (1Ø) 	0.2kW	0.2kW	15A	NEMA 5-15P



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	40
5.2 Installation Tools and Items	41
5.3 Move the Pallet	42
5.4 Unpack the Machine Crate	43
5.5 Install Core Components	47
5.6 Verify the Installation	52
5.7 Install Accessory Components	56
5.8 Set Up the PathPilot Controller	63

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.

- Dead-blow hammer (or similar)
- Metric hex wrench set
- Pallet jack
- Phillips screwdriver
- Safety eyewear that meets ANSI Z87+
- Snips
- Work gloves

5: INSTALLATION

5.3 Move the Pallet

5.3 MOVE THE PALLET

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



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

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



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

5.4 UNPACK THE MACHINE CRATE

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

1. Put on work gloves and eye protection.
2. Remove the screws from the top panel of the shipping crate with a cordless drill and a Phillips bit. With an assistant, slide the top panel off of the crate and set it aside.



Figure 5-1: Removing the top panel from the shipping crate.

3. Remove the screws on each of side of the crate and, with an assistant, lift the side of the crate off of the pallet. Work your way around the crate until all four sides are removed.



Figure 5-2: Removing the screws from the side panels of the shipping crate.

4. Remove and discard the plastic wrapped around the machine and stapled to the pallet.
5. Inspect the item(s):

- Photograph any damage that may have occurred during shipping.
- Verify the received goods against the packing list. If there is any damage or shortages, you must contact Tormach within 30 days of receipt. Create a support ticket with Tormach Technical Support at tormach.com/how-to-submit-a-support-ticket for guidance on how to proceed.

5: INSTALLATION

5.4 Unpack the Machine Crate

6. Several items are strapped in place to prevent shipping damage. Remove them as follows:
 - a. Remove and discard the shipping straps securing the operator console's arm to the enclosure.
 - b. Identify the red shipping bracket on the operator console's arm. Remove the bracket with a hex wrench, and reinstall the screws in their original location. Set the bracket aside for future use.

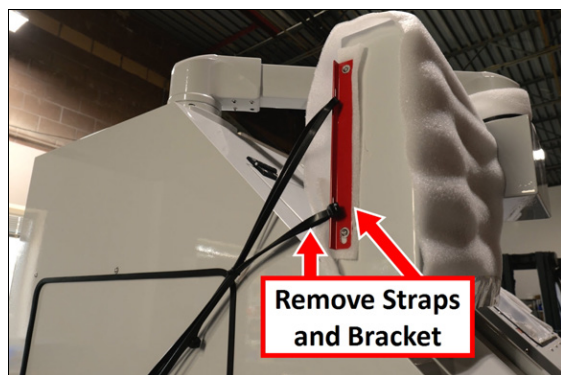


Figure 5-3: Operator console arm shipping straps and bracket.

- c. From the enclosure's right side, remove the side access panel. Set aside the panel and all of its screws.



Figure 5-4: Enclosure right side access panel.

- d. From the inside of the machine, identify the red shipping bracket securing the carriage to the bed casting. Remove the bracket, and set it aside, along with its screws, for future use.

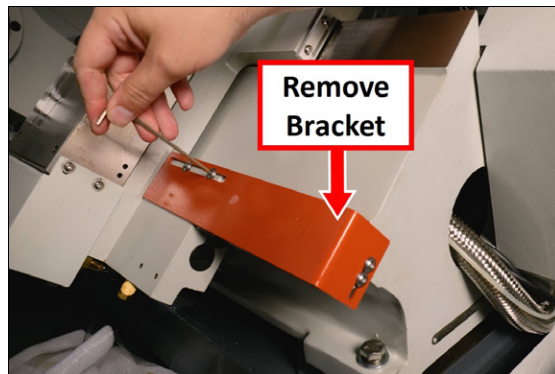


Figure 5-5: Carriage shipping bracket.

- e. Identify the red shipping bracket securing the enclosure door. Remove the bracket and reinstall the screws in their original location. Set the bracket aside for future use.

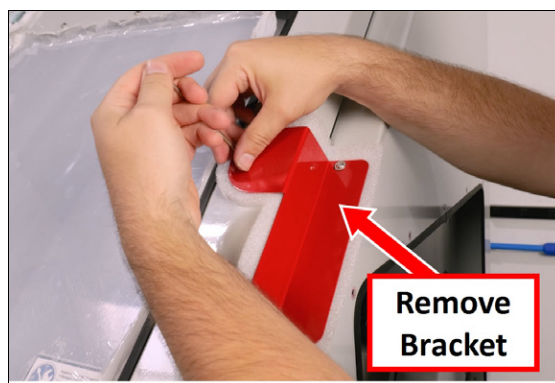


Figure 5-6: Enclosure door shipping bracket.

- f. Identify the remaining shipping bracket on the enclosure door. Remove and discard the shipping straps, then remove the bracket and reinstall the screws in their original location. Set the bracket aside for future use.



Figure 5-7: Enclosure door shipping bracket.

- g. Remove and discard the remaining shipping straps securing the items on the pallet.
- 7. Identify the operator console box on the machine's pallet, and set it aside. Open the enclosure door, and remove and set aside the tool bag and any remaining packages that shipped in the machine.
- 8. From the tool bag, identify and remove the machine's feet. Set them aside until later in this procedure.
- 9. Slide the coolant tank from out of the machine. Remove and set aside the chip tray, and then move the coolant tank off of the pallet. Set it aside with the chip tray.
- 10. To remove the pallet, you'll need to raise the machine from the back with a standard pallet jack. Verify that there's clearance behind the machine to move the jack in and out. If there's not, lift the pallet from the side where the coolant tank slides out and reposition it.
- 11. Disassemble and remove the pallet as follows:
 - a. Remove the six bolts along the back of the pallet with a 9/16 in. (15 mm) socket wrench. Remove the board from the pallet.

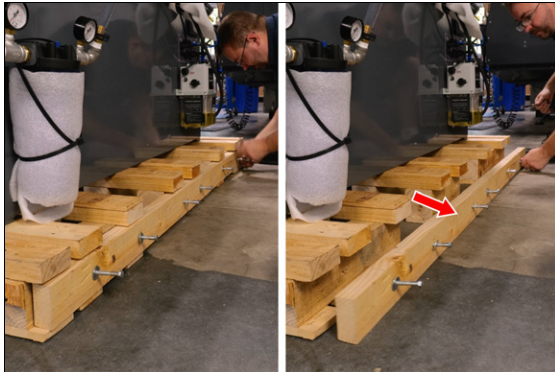


Figure 5-8: Removing the pallet's rear board.

- b. The machine is secured to the pallet from below. Use a 1-13/16 in. (30 mm) socket wrench to remove the four screws from each corner of the machine.

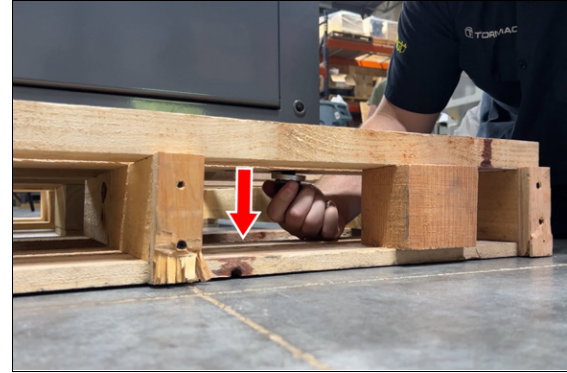


Figure 5-9: Removing the bolts securing the machine to the pallet.

- c. Slowly lift the machine up with a pallet jack until it's clear of the pallet. Slide the pallet out from the front and set it aside.



Figure 5-10: Removing the pallet from under the machine.

- 12. Identify the four machine feet (threaded rods and foot pads) that you set aside earlier.
- 13. Thread an M20 nut onto each leg, followed by an M20 washer. Position the nut so that it's near the bottom of the threads.
- 14. If your pallet jack doesn't lift the machine high enough to provide clearance to install the assembled machine foot, remove each foot pad from its socket.

5: INSTALLATION

5.4 Unpack the Machine Crate

15. With the machine still lifted up by the pallet jack, install the threaded rod onto each corner so that the distance from the bottom of the stand to the bottom of the threaded rod's socket is approximately 3 in. (76 mm).

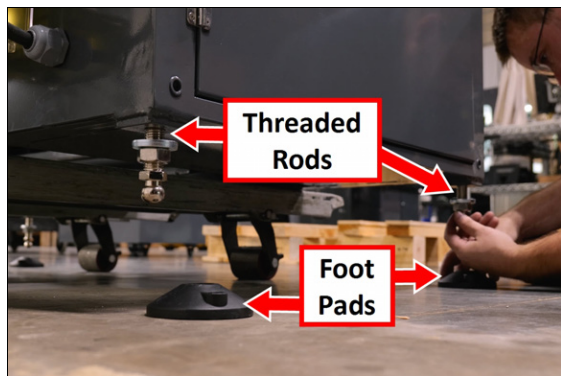


Figure 5-11: Installing the threaded rods into each corner of the machine.



WARNING! Crush Hazard: Make sure to lower the machine slowly and evenly to avoid sudden movements that could cause injury or damage the machine.

16. Align the foot pads under each corner of the machine. Lower the machine with the pallet jack until the threaded rods are seated in the foot pads. The foot pads will snap onto the ends of the threaded rods as the machine lowers.



Figure 5-12: Aligning the threaded rod with the foot pad.

17. Raise the pallet jack back up enough to elevate the machine, and adjust each machine foot so that it's 3 in. (76 mm) from the bottom of the foot pad to the bottom of the machine stand.

18. Slowly lower the pallet jack until the machine is completely supported by its feet. Once the machine is safely on the ground, verify that all four feet are in contact with the ground. Adjust any feet as needed until the machine is stable and secure, and then remove the pallet jack.
19. Tighten the nuts and washers firmly against the underside of each corner of the machine to lock the feet in place.

5.5 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:

5.5.1 Assemble the Coolant Tank.....	47
5.5.2 Install the PathPilot Operator Console.....	47
5.5.3 Install the Air System Components.....	48
5.5.4 Set Up the Automatic Oiler.....	50

5.5.1 Assemble the Coolant Tank

1. From the tool bag, identify the four casters and hardware for the coolant tank. Stand the coolant tank on end to install the casters.

Note: We recommend standing the coolant tank onto a piece of protective packaging material to prevent scratches to the paint.

2. Install the four casters onto the coolant tank with four M6 × 1 - 8 mm socket head cap screws, four M6 lock washers, and four M6 flat washers.



Figure 5-13: Installing casters on the coolant tank.

3. Set the coolant tank down onto its wheels.
4. Connect the coolant hose to the brass fitting on the coolant pump, and connect it with a hose clamp and pliers.



Figure 5-14: Connecting the coolant hose to the coolant pump.

5. Route the loose end of the coolant hose to the brass fitting on the pre-installed coolant filter on the back of the machine stand, and connect it with a hose clamp and pliers.



Figure 5-15: Connecting the coolant hose to the coolant filter.

6. Connect the power cord from the coolant pump to the outlet of the cable in the track that's connected to the **Flood Coolant** power outlet on the back of the machine.
7. Place the chip basket into the coolant tank, and then roll the coolant tank into the machine stand.

5.5.2 Install the PathPilot Operator Console

1. Identify the operator console box that you set aside while unpacking the machine crate. Open the box and remove the operator console and its mounting hardware.

5: INSTALLATION

5.5 Install Core Components

2. Remove and set aside the back panel of the operator console mount.



Figure 5-16: Operator console mount's back panel.

3. Remove and discard the cable ties securing the operator console's cables.
4. With an assistant, install the operator console onto the mount.

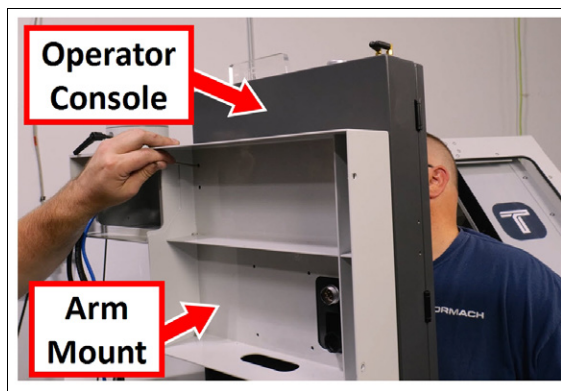


Figure 5-17: Mounting the operator console onto the arm.

5. Connect the loose end of the emergency stop cable to the **Emergency Stop** outlet on the back of the operator console.

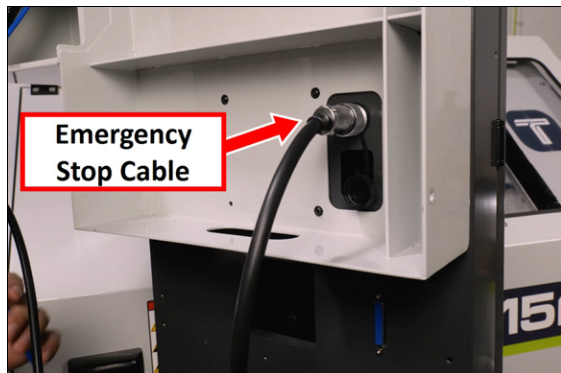


Figure 5-18: Emergency Stop cable connected to the operator console.

6. Connect the male end of the power cable to the AC adapter, and set it inside the sheet metal housing.
7. Route the power cable and the Ethernet cable through the hole on the bottom of the operator console mount and into the hole on the back of the operator console.
8. Remove the three screws on the operator console's door with a 3 mm hex wrench. Set the screws aside.
9. Open the operator console door, and route the loose end of the Ethernet cable to the LAN1 outlet on the console. Then, route the loose end of the power cable to the DC power outlet on the console.

Note: If you plan to use the machine with a wired Ethernet connection, connect the cable to the LAN2 outlet on the console.

10. Install the mouse tray the right side of the operator console. Place the mouse and the keyboard onto their trays, and route the loose end of the cables to the USB outlets on the inside of the operator console.
11. Close the operator console door and secure it with the screws that you set aside earlier.
12. Attach the jog pendant bracket to the right side of the operator console. Then, connect the jog pendant to the outlet on the back of the operator console, and use the magnet in the pendant to mount it to the bracket.
13. Reinstall the operator console's mount panel that you set aside earlier.
14. Insert the operator console's key on the top of the console.

5.5.3 Install the Air System Components

1. Install the air gun holster on to the front of the lathe with two screws and a hex wrench.



Figure 5-19: Installing the air gun holster.

2. Plug the air gun into the coiled air hose, and place the gun into the holster.
3. Remove the screws securing the left side access panel on the enclosure. Set aside the screws and the panel.
4. From the front of the enclosure, remove the two knockout panels on the left side of the enclosure with a hammer and punch. The upper hole is used for the pressure regulator, and the lower hole is used for the female-to-female air fitting.



Figure 5-20: Removing the knockout panels on the front of the enclosure.

5. Install the female-to-female air fitting in the lower hole. Use two wrenches to tighten the fitting securely.

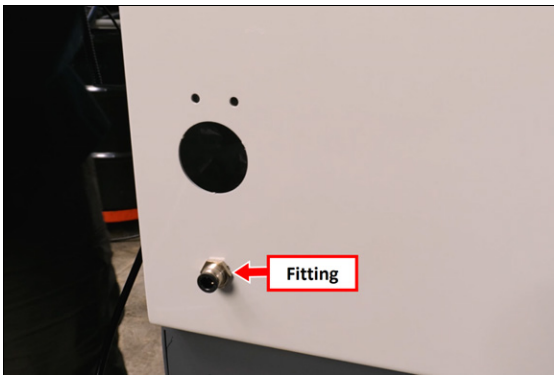


Figure 5-21: Female-to-female air fitting installed.

6. Insert the loose end of the coiled air hose from the air gun into the push-to-connect fitting on the installed female-to-female connection.
7. Remove the red threaded ring from the pressure regulator, and then slide the mounting bracket onto the regulator body and reinstall the red threaded ring.

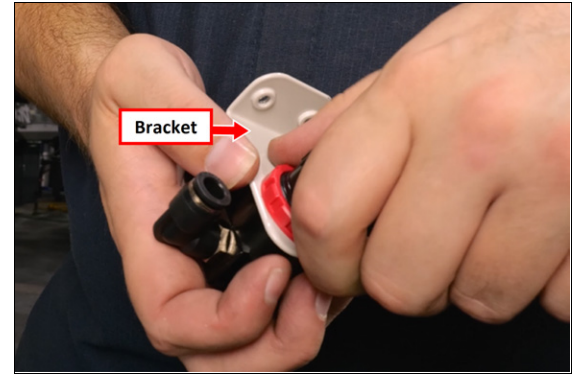


Figure 5-22: Assembling the pressure regulator onto its bracket.

8. Position the regulator into the upper hole and secure the bracket with two screws.



Figure 5-23: Installing the pressure regulator.

9. Install the pressure regulator panel onto the rear right side of the enclosure with the provided screws.



Figure 5-24: Installing the pressure regulator panel.

5: INSTALLATION

5.5 Install Core Components

10. Remove one of the regulator's push-to-connect plugs, and insert one end of the long, thin air line into the open fitting.



Figure 5-25: Air line connected to the pressure regulator panel.

11. Route the loose end of the long air line through the rear access panel and into the left cabinet of the enclosure. Inside the enclosure, route it along the existing cable path. Secure it to the cables with cable ties.

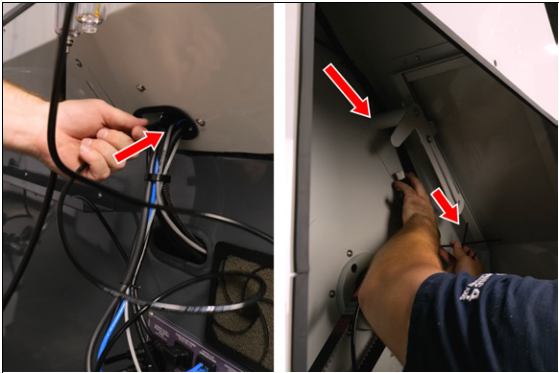


Figure 5-26: Routing the long air line to the pressure regulator.

12. Connect the loose end of the thin air line to the right push-to-connect fitting on the regulator. Connect the short, thicker air line to the left push-to-connect fitting on the regulator. Connect the loose end to the inlet adapter below.

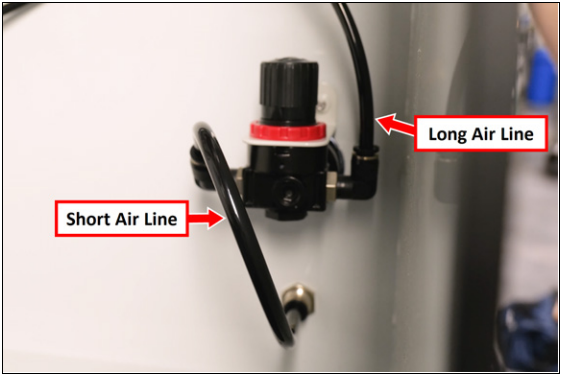


Figure 5-27: Air line connections made at the pressure regulator.

13. Verify that all air fittings and connections are tight and secure.

5.5.4 Set Up the Automatic Oiler

Complete the following steps in the order listed:

Specify the Interval Time	50
Specify the Actuation Time	51

Specify the Interval Time

The oiler distributes oil at the following times:

- When the machine is powered on
- After a specified interval

The interval time is the amount of time, in minutes, that the oiler waits between oil applications.

Note: If your machine will be unused for a long period of time (like overnight), we recommend powering off the machine to avoid over-lubricating the system.

1. On the oiler's control panel, push and hold either of the **Minutes Adjustment** buttons. It doesn't matter which button you push.

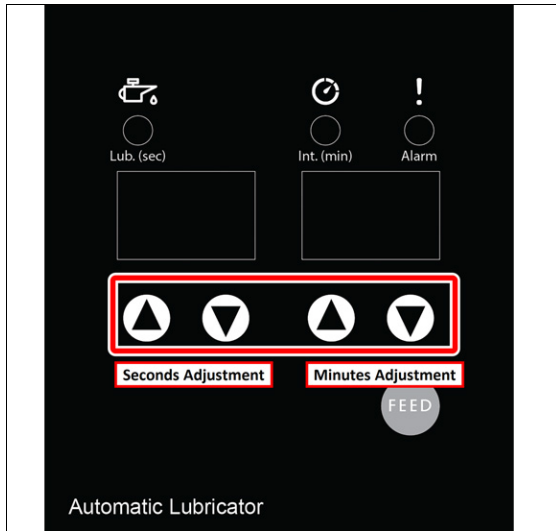


Figure 5-28: Automatic oiler control panel.

The oiler beeps.

2. While looking at the **Minutes** display, do one of the following:
 - To increase the interval time, push the **Up Arrow**.
 - To decrease the interval time, push the **Down Arrow**.

NOTICE! Pressure Hazard: To avoid potential equipment damage, don't set the interval time to less than five minutes.

Specify the Actuation Time

The actuation time is the amount of time, in seconds, that the oiler distributes oil during an oil application.

1. On the oiler's control panel, push and hold either of the **Seconds Adjustment** buttons. It doesn't matter which button you push.
The oiler beeps.
2. While looking at the **Seconds** display, do one of the following:
 - To increase the interval time, push the **Up Arrow**.
 - To decrease the interval time, push the **Down Arrow**.

NOTICE! Pressure Hazard: To avoid potential equipment damage, don't set the actuation time to more than three minutes.

5: INSTALLATION

5.6 Verify the Installation

5.6 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:

5.6.1 Power on the Machine	52
5.6.2 Verify Axes Function	53
5.6.3 Verify Spindle Function	54
5.6.4 Power off the Machine	55

5.6.1 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 230 Vac, 50/60 Hz
 - **Recommended Circuit Amperage** 30 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 5-29: The Main Disconnect switch on the side of the machine's electrical cabinet.

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

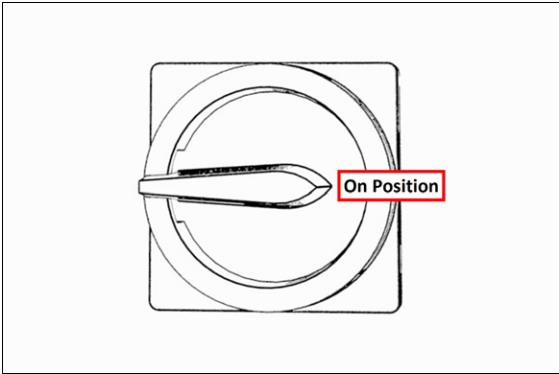


Figure 5-30: Example of the Main Disconnect switch in the On position.

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. 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.

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



Figure 5-31: Emergency Stop button.

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



Figure 5-32: Reset button.

The axis drives are now powered on.

9. 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.

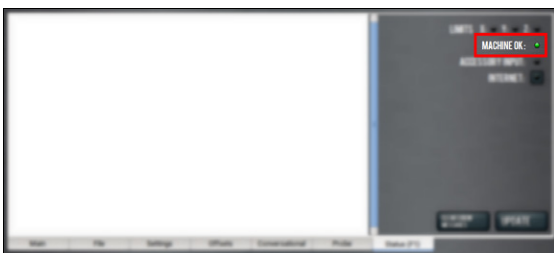


Figure 5-33: Machine OK light on the Status tab.

10. Select **Reset**.



Figure 5-34: Reset button.

This initializes the connection between the machine and controller.

5.6.2 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 5-35: 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-).

5: INSTALLATION

5.6 Verify the Installation

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.

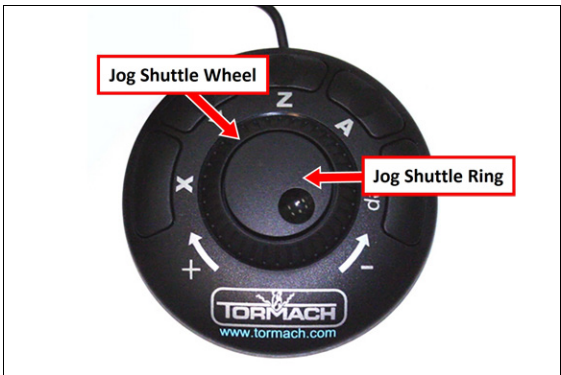


Figure 5-36: Functions on the optional Jog Shuttle.

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

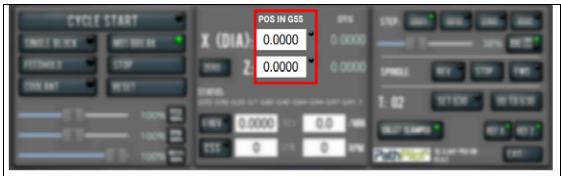


Figure 5-37: Axis lights.

- 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.



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 119).

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**.

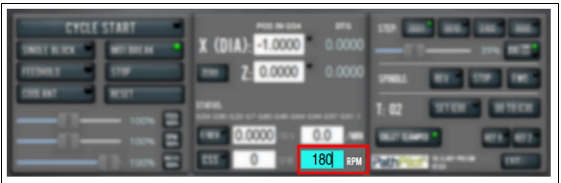


Figure 5-39: Spindle controls.

The spindle rotates clockwise at 180 rpm.

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

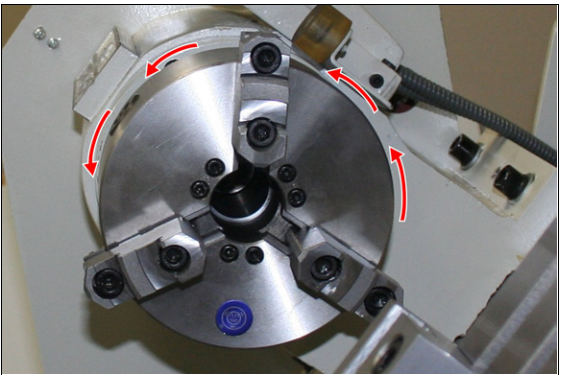


Figure 5-40: Spindle rotating counterclockwise, in the correct forward direction.

5. Select **STOP**.
The spindle stops rotating.
6. Select **REV**.
The spindle rotates clockwise at 180 rpm.

5.6.3 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 5-38: Spindle Override slider.

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

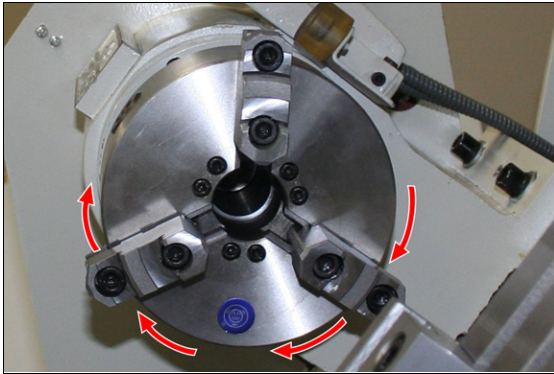


Figure 5-41: Spindle rotating clockwise, in the correct reverse direction.

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

5.6.4 Power off the Machine

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



Figure 5-42: 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**.

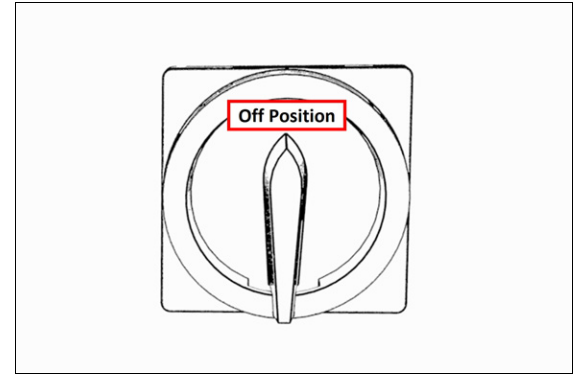


Figure 5-43: Example of the Main Disconnect switch in the Off position.

Mains power is disconnected from the machine.

5: INSTALLATION
5.7 Install Accessory Components

5.7 INSTALL ACCESSORY COMPONENTS

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

5.7.1 Install the Automatic Collet Closer for 15LX.56
5.7.2 Connect the Collet Closer Foot Pedal Kit. 62

5.7.1 Install the Automatic Collet Closer for 15LX

Complete the following steps in the order listed:

Required Tools.56
Before You Begin. 56
Mount the Collet Closer Cylinder Assembly. 56
Mount the Anti-Rotation Bracket and Solenoid. 58
Make Air Connections.59
Make Electrical Connections.60
Align the Automatic Collet Closer Kit with the Spindle. 60
Operation. 61
Automatic Operation. 61
Manual Operation. 61
Switch Between ID and OD Clamping.61
Change the Collet.61
Adjust the Holding Force.62
Specifications. 62

Required Tools

Before you begin, make sure you have the following tools and items:

- 3/16 in. hex wrench
- 2.5 mm hex wrench
- 3 mm hex wrench
- 10 mm wrench
- Dead-blow hammers, two (or similar)
- Magnetic dial indicator
- Ratchet with 3/16 in. hex bit
- Round stock

Before You Begin

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

2. Disconnect the compressed air supply, and wait for any residual air pressure in the system to release.

Mount the Collet Closer Cylinder Assembly

1. Remove the eight screws securing the spindle compartment access panel with a 3 mm hex wrench. Set aside the screws and the access panel.

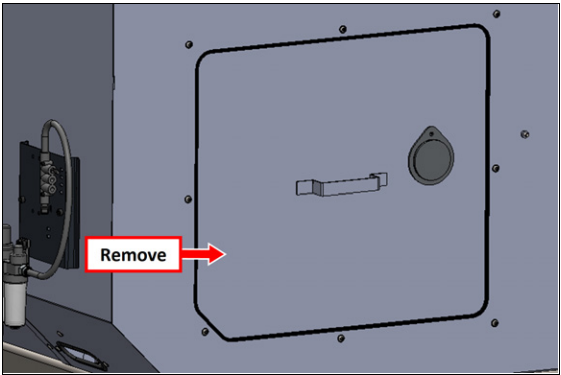


Figure 5-44: Access panel.

2. Find the M6 x 1 - 20 flanged hex head screws provided. Mount the adapter spacer onto the spindle pulley with four M6 screws and a 10 mm wrench.

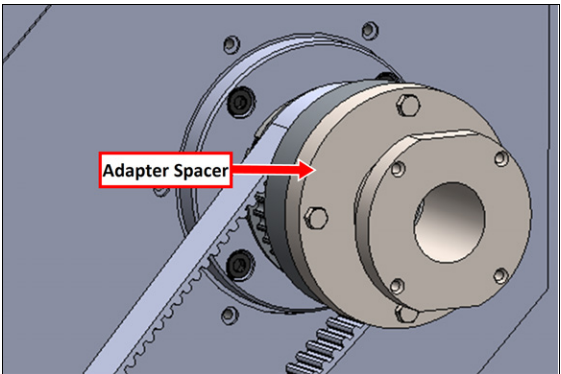


Figure 5-45: Adapter spacer mounted onto the spindle pulley.

3. Find the draw tube provided. One end of the draw tube has four holes, which are used to secure the draw tube to the pneumatic cylinder.

Note: Installation steps differ depending on machine serial number. If your machine is serial number LB10061 or later, go to Step 4. If your machine is serial number LB10060 or earlier, go to Step 5.

4. Machines LB10061 and Later

- a. Assemble the draw tube with the pneumatic cylinder so that the draw tube's holes are furthest away from the pneumatic cylinder, as shown in the following image.

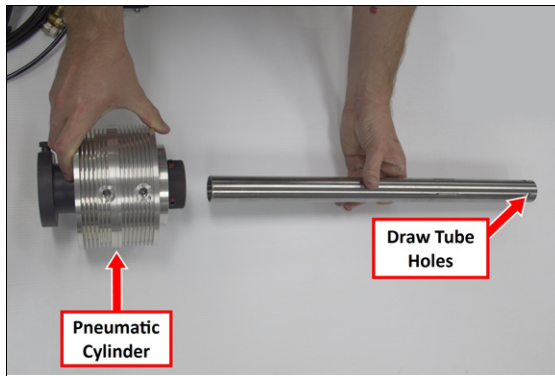


Figure 5-46: Draw tube aligned with a pneumatic cylinder.

- b. Put the draw tube through the pneumatic cylinder until the end of the draw tube is flush with the end of the pneumatic cylinder.

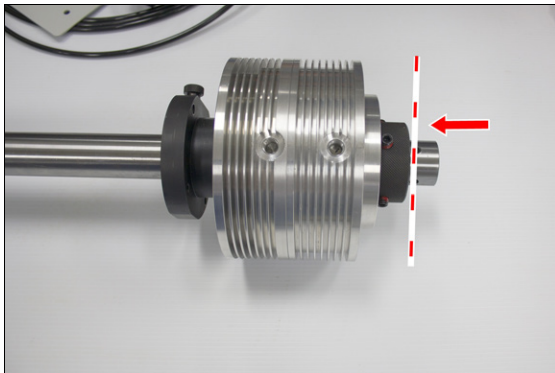


Figure 5-47: Draw tube in the pneumatic cylinder.

- c. Use a 3/16-in. hex wrench to tighten the four set screws on the pneumatic cylinder that are aligned with the four holes on the draw tube. The draw tube is now secured in place.
- d. Slowly slide the pneumatic cylinder assembly all the way into the spindle. Rotate the cylinder so that its flats align with the flats on the adapter spacer.
- e. Go to Step 6.

5. Machines B10060 and Earlier

- a. Partially assemble the draw tube with the pneumatic cylinder so that the draw tube's holes are furthest away from the pneumatic cylinder, as shown in the following image. Keep approximately 3 in. of the tube sticking out from the mounting flange of the cylinder.

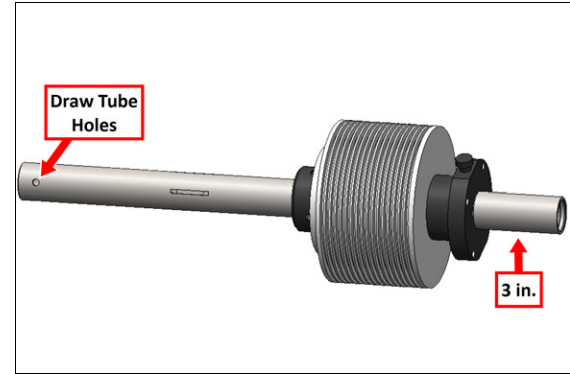


Figure 5-48: Draw tube partially assembled.

- b. Maneuver the air cylinder around the wall of the enclosure compartment, and then insert the end of the draw tube into the spindle.

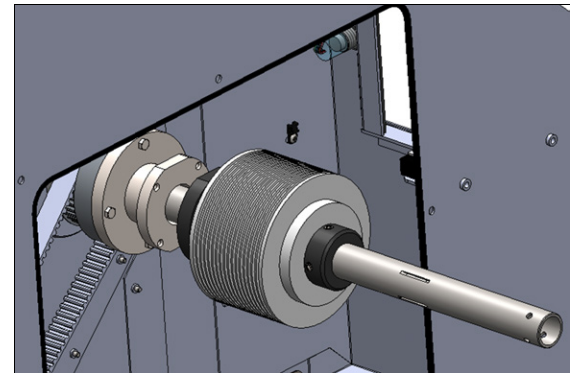


Figure 5-49: Draw tube inserted into the spindle.

- c. While keeping the draw tube in the spindle, slide the draw tube the rest of the way into the pneumatic cylinder until the end of the draw tube is flush with the end of the pneumatic cylinder.
- d. Use a 3/16-in. hex wrench to tighten the four set screws on the pneumatic cylinder that are aligned with the four holes on the draw tube. The draw tube is now secured in place.
- e. Go to Step 6.

5: INSTALLATION

5.7 Install Accessory Components

6. Mount the cylinder to the adapter spacer with four M6 × 1 - 25 hex head screws. Secure the screws finger-tight for now.

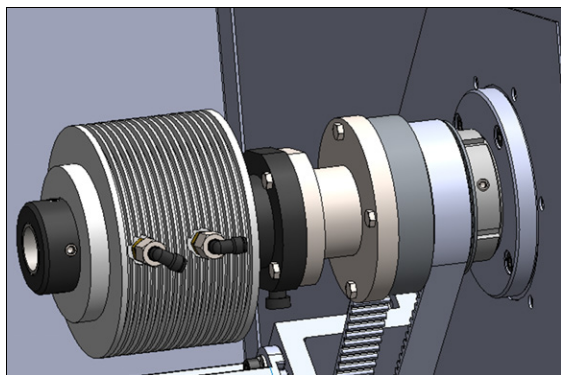


Figure 5-50: Pneumatic cylinder mounted to the adapter spacer.

7. Loosen the thumb screw on the pneumatic cylinder assembly. Loosening the thumb screw allows the draw tube to rotate separately from the spindle so that you can thread a collet into the draw tube.
8. Verify that a 5C insert is installed in the spindle, as shown in the following image.

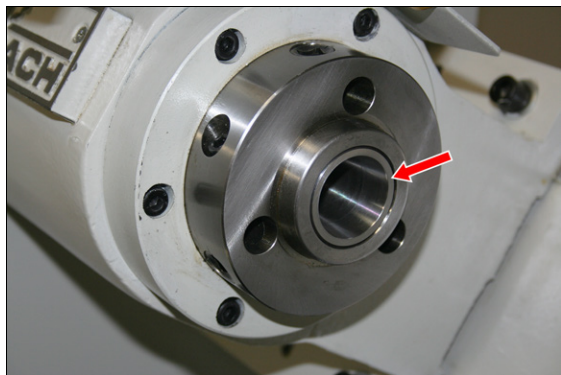


Figure 5-51: 5C insert installed in the spindle.

9. Install a collet into the 5C insert. You must rotate the collet until its groove aligns in the 5C insert.



Figure 5-52: Collet installed into the 5C insert.

10. Put a piece of stock into the collet.
11. Hold the collet with one hand and then, from the pneumatic cylinder, use your other hand to turn the draw tube handle.
12. Continue to thread the collet on the draw tube until it is completely threaded.
13. Tighten the thumb screw on the pneumatic cylinder assembly.
14. To prevent the draw tube from turning during operation, you must make sure that the thumb screw is engaged in the draw tube: Turn the draw tube handle counterclockwise until the thumb screw clicks into one of four slots on the draw tube.
The draw tube is now locked into place.

Mount the Anti-Rotation Bracket and Solenoid

1. Find the M3 × 0.5 - 30 socket head cap screws, the solenoid valve assembly, and the anti-rotation bracket.
2. Mount the solenoid valve assembly to the anti-rotation bracket with two M3 screws and a 2.5 mm hex wrench.

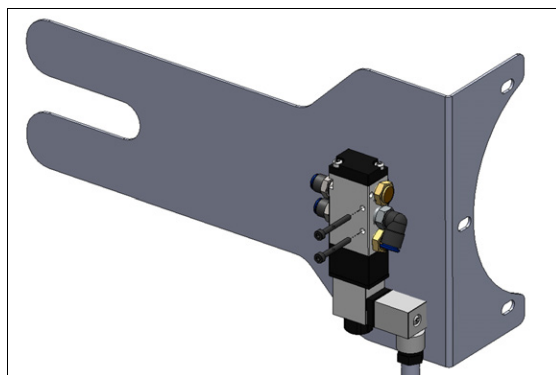


Figure 5-53: Mounting the solenoid assembly to the anti-rotation bracket.

- Identify the mounting location for the anti-rotation bracket as shown in the following image.

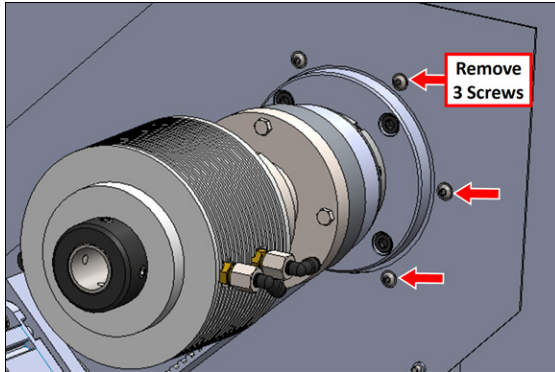


Figure 5-54: Mounting location for the anti-rotation bracket.

- Remove three flanged button head cap screws from the enclosure with a 3 mm hex wrench and set the screws aside.
- Align the slot in the anti-rotation bracket with the fittings in the pneumatic cylinder, and mount the bracket to the enclosure with the three screws that you removed in the previous step.

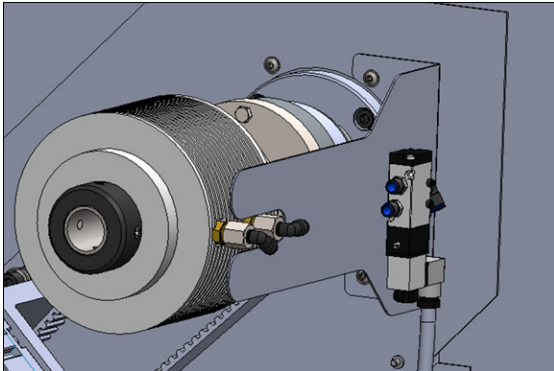


Figure 5-55: Anti-rotation bracket mounted.

Make Air Connections

- Find the 1/4 in. tubing provided. Connect the fittings on the solenoid valve to the fittings on the pneumatic cylinder with the two shorter pieces of tubing.

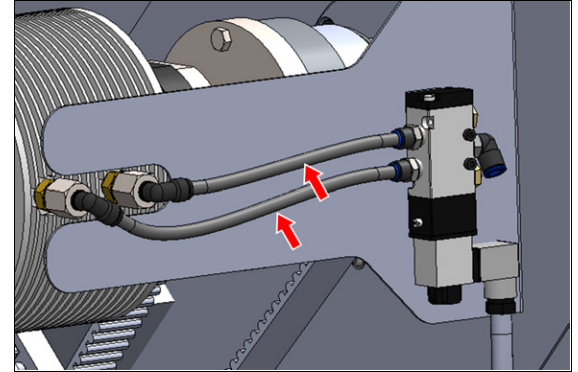


Figure 5-56: Air connections between the solenoid valve and the pneumatic cylinder.

- Connect the longer piece of 1/4 in. tubing to the elbow fitting on the solenoid valve.

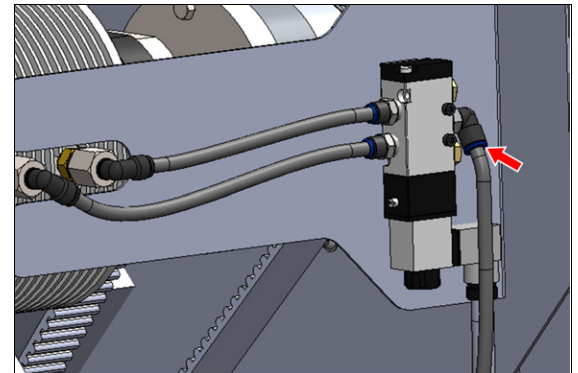


Figure 5-57: Longer 1/4 in. tubing connected to the elbow fitting.

- Route the loose end of the tubing toward the back of the spindle compartment and out of the access slot.
- Connect the loose end of the tubing to any available port on the pneumatics manifold assembly.

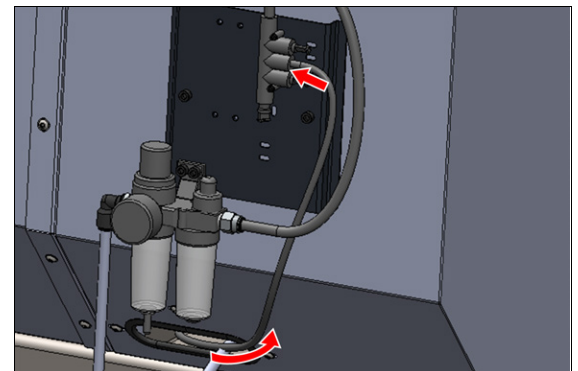


Figure 5-58: Longer 1/4 in. tubing connected to the pneumatics manifold assembly.

5: INSTALLATION

5.7 Install Accessory Components

Make Electrical Connections

1. Route the loose end of the power cable from the solenoid toward the back of the spindle compartment and out of the access slot.
2. Connect the cable to the **Accessory Output 2** outlet on the back of the machine stand.



Figure 5-59: Electrical connections panel on the back of the machine stand.

Align the Automatic Collet Closer Kit with the Spindle

1. Connect the quick-connect fitting to the air compressor.



Note: You may hear the collet closer clunk. Connecting the shop air supply moves the cylinder into the clamped (closed) position.

2. Put the base of a magnetic dial indicator on the top of the belt guard cover. Verify that the magnetic dial indicator touches the aluminum of the pneumatic cylinder furthest away from the belt guard cover, as shown in the following image.

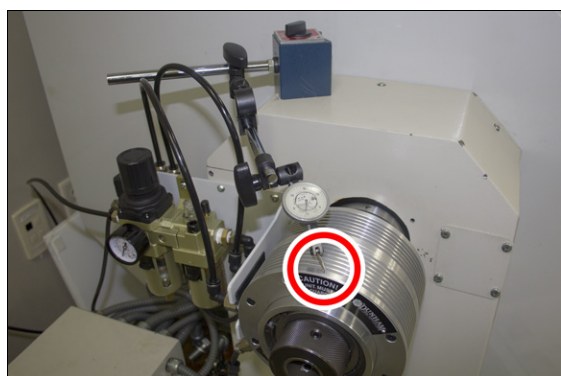


Figure 5-60: Magnetic dial indicator put on a pneumatic cylinder.



Note: We don't recommend mounting the dial indicator base to the sheet metal of the enclosure. The sheet metal can flex, which impacts the results of the alignment.

3. Manually rotate the spindle clockwise. On the magnetic dial indicator, the needle rises and falls. Continue to rotate the spindle until the needle reaches the highest point on the magnetic dial indicator.
4. Gently tap the mounting flange with two dead-blow hammers, as shown in the following image.

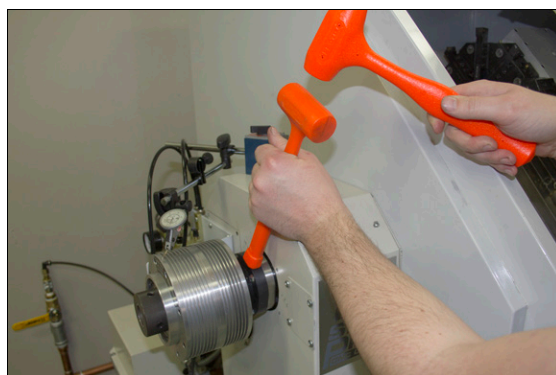


Figure 5-61: Using two dead-blow hammers to align an Automatic Collet Closer Kit.

5. Manually rotate the spindle clockwise until the needle again reaches the highest point on the magnetic dial indicator.
6. Continue adjusting until the runout is less than 0.002 in.



Note: If the reading on the magnetic dial indicator remains the same after you tap the mounting flange with the dead-blow hammer, slightly loosen the four M6 × 1 - 25 hex head screws, and repeat the alignment steps.

7. Tighten the four M6 × 1 - 25 hex head screws on the mounting flange with a 10 mm wrench.



Figure 5-62: Tighten the mounting flange on the pneumatic cylinder.

8. Manually rotate the spindle to verify that the runout is still less than 0.002 in. If it's not, repeat the alignment steps.
9. 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).

You have completed installing the Automatic Collet Closer Kit. For operating instructions, go to "Operation" (below).

Operation

WARNING! Operator Knowledge: You and all other operators must read and understand the 15LX operator manual before installing, using, or maintaining the machine.

Automatic Operation

- Use M-code to program the collet closer to open and close:
 - **M10** opens the collet closer.
 - **M11** closes the collet closer.

Manual Operation

- From the PathPilot interface, select **Collet Clamped** to toggle the button's light on and off.

When the **Collet Clamped** button's green light is on, the collet is in the closed (clamped) position. When the **Collet Clamped** button's green light is off, the collet is in the open (unclamped) position.



Figure 5-63: Collet Clamped button with the light on.

Switch Between ID and OD Clamping

To switch between clamping the inside diameter and the outside diameter of a workpiece:

- From the PathPilot interface, on the **Settings** tab, select either the **ID Clamping** or the **OD Clamping** option.

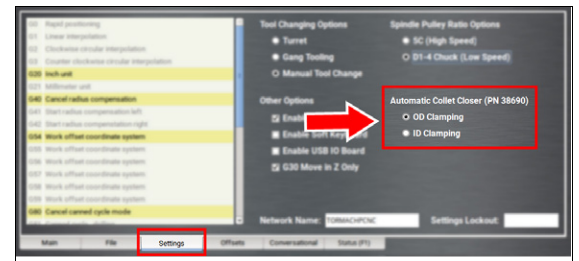


Figure 5-64: Automatic Collet Closer Kit clamping options.

Change the Collet

1. From the PathPilot interface, select **Collet Clamped** to toggle the button's light off.



Figure 5-65: Collet Clamped button's light off.

The collet closer is now in the open position.

2. Loosen the thumb screw on the pneumatic cylinder assembly. Loosening the thumb screw allows the draw tube to rotate separately from the spindle so that you can thread a collet into the draw tube.

5: INSTALLATION

5.7 Install Accessory Components

- 3. Verify that a 5C insert is installed in the spindle, as shown in the following image.

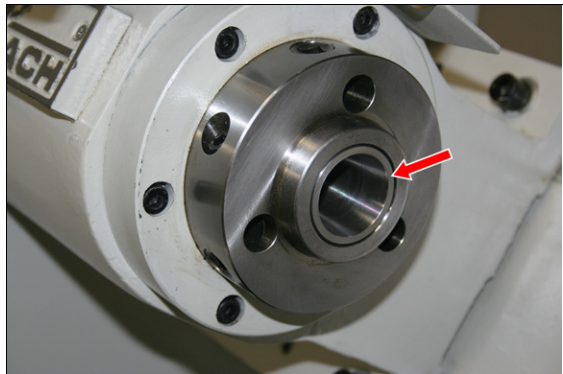


Figure 5-66: 5C insert installed in the spindle.

- 4. Install a collet into the 5C insert. You must rotate the collet until its groove aligns in the 5C insert.



Figure 5-67: Collet installed into the 5C insert.

- 5. Hold the collet with one hand and then, from the pneumatic cylinder, use your other hand to turn the draw tube handle.
- 6. Continue to thread the collet on the draw tube until it is completely threaded.
- 7. Tighten the thumb screw on the pneumatic cylinder assembly.
- 8. To prevent the draw tube from turning during operation, you must make sure that the thumb screw is engaged in the draw tube: Turn the draw tube handle counterclockwise until the thumb screw clicks into one of four slots on the draw tube.
The draw tube is now locked into place, and the collet is ready for the workpiece.

Adjust the Holding Force

If needed, you can adjust the holding force using the pressure regulator:

- Increase the pressure to increase the holding force.
- Decrease the pressure to decrease the holding force.



Note: Refer to "Specifications" (below) for information on holding force limits.

We recommend you begin with 40 psi, and then gradually increase the pressure until the part is sufficiently held.

Specifications

Maximum Continuous rpm	3600 rpm
Maximum psi	100 psi
Pull Force	1600 lb at 100 psi
Push Force	600 lb at 100 psi

5.7.2 Connect the Collet Closer Foot Pedal Kit

- Route the foot pedal cord to the back left corner of the machine (near the electrical cabinet). Connect it to the **Accessory Inputs 1-2** port.

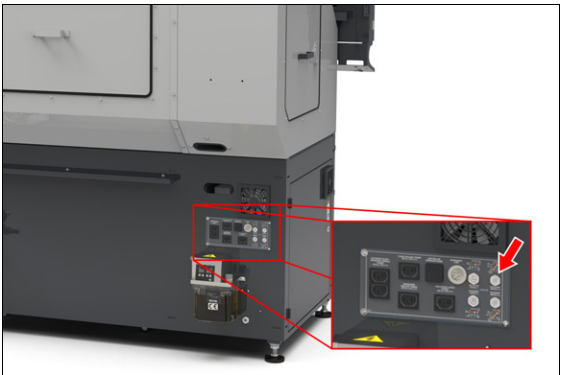


Figure 5-68: Accessory Output 1-2 port on the machine stand.

5.8 SET UP THE PATHPILOT CONTROLLER

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

5.8.1 Specify the Date and Time

1. If you haven't yet done so, turn the key on the top of the operator console into the setup mode position. You can identify which mode the controller is in by referencing the **Status** read-only field.



Figure 5-69: PathPilot indicates the controller's current mode.

2. 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.

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

5.8.2 Specify the Keyboard Language

By default, the keyboard language is set to English.

To specify a different keyboard language:

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

5.8.3 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**.

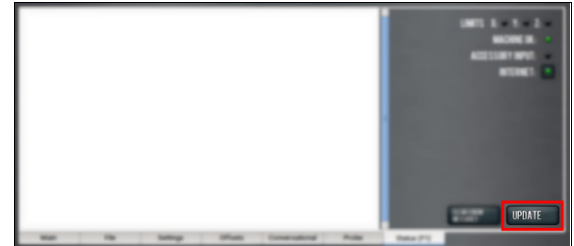


Figure 5-70: Update button on the Status tab.

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

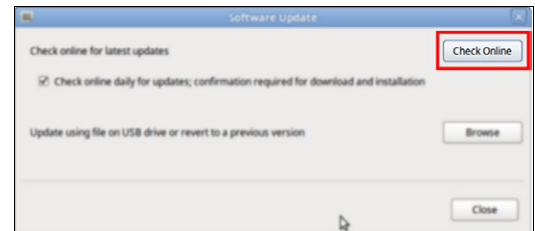


Figure 5-71: Software Update dialog box.

4. Select **Install**.

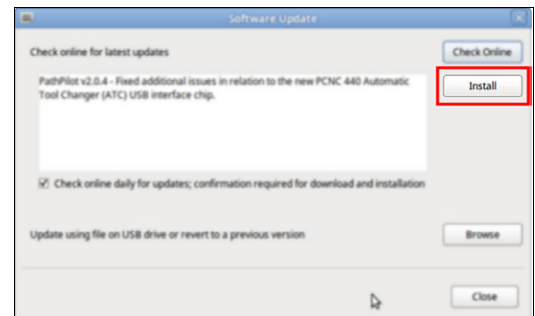
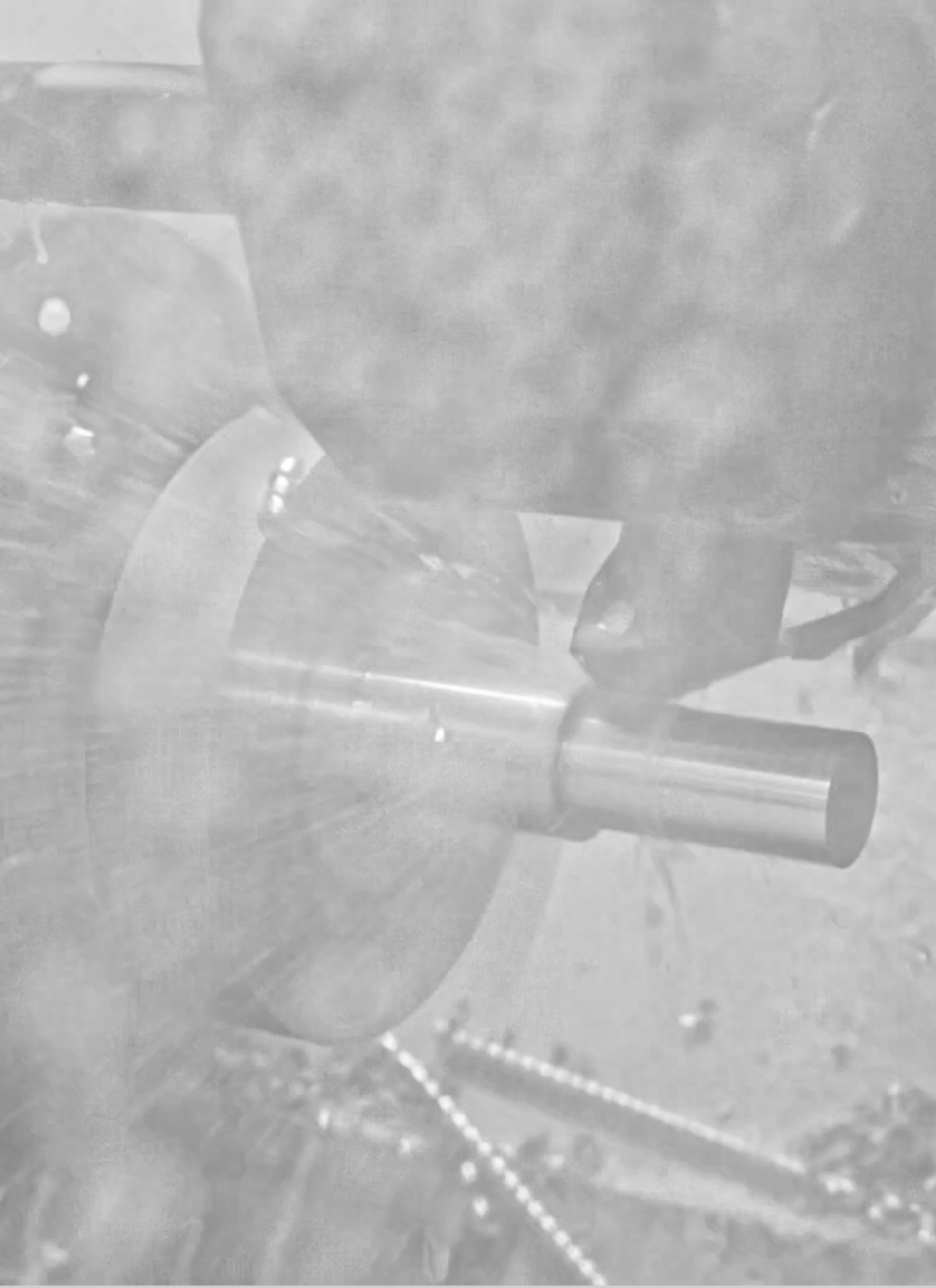


Figure 5-72: Install button on the Software Update dialog box.

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	66
6.2 Tooling Reference	67
6.3 Basic Controls Reference	69



TORMACH®

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 Cross Slide Table

The cross slide table is 26" x 6" (660 mm x 152 mm). The cross slide table has ten M8 threaded holes that you can use to mount gang tooling plates and a tool turret. Always verify that cutting tools and tool holding devices are properly secured before starting a cutting operation.

6.1.2 Spindle

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

1. A 5C taper insert used with 5C collets, ideal for precise clamping of small-diameter parts.
2. Chucks with D1-4 cam lock mounting, suitable for holding larger or irregularly shaped stock.

- **Spindle Power**

- **Continuous** 6 hp (4.5 kW)
- **Peak** 10 hp (7.4 kW)

- **Spindle Bore Diameter** 1.53" (38.86 mm) (with 5C insert removed)

About the Spindle

On a CNC lathe, the spindle holds and rotates the workpiece, while the cutting tools remain stationary on the cross slide table. Material is removed as the rotating stock is brought into contact with the cutting tools.

The spindle is driven by a motor through a timing belt, providing the torque required for various turning operations. Ensure the appropriate workholding device is securely installed before starting spindle operations. Operate the spindle either manually or by G-code commands (entered in the MDI Line DRO field or programmed into a G-code program). Manually controlling the spindle is useful during setup procedures. The spindle can rotate in either direction clockwise (reverse) or counterclockwise (forward) at a specified spindle speed.

6.1.3 Axes

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

- The Z-axis, which is parallel to rotational axis of the spindle (along the length of the bed casting).

The -Z direction is toward the headstock, and the +Z direction is away from the headstock.

- The X-axis, which is perpendicular to the Z-axis and nominally parallel to the long dimension of the cross slide table.

The -X direction is toward the front (toward the operator), and +X direction is toward the rear (away from the operator).

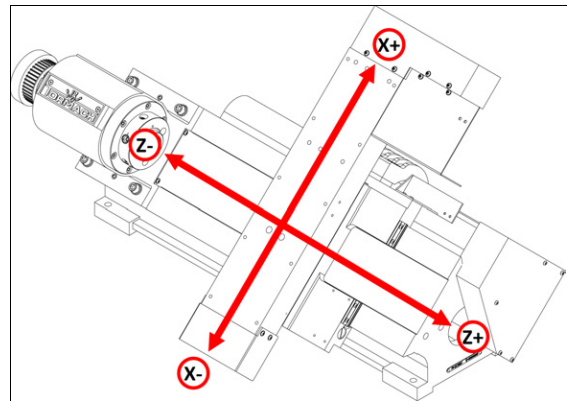


Figure 6-1: Axes directions on the machine.

- **X-Axis** 10" (254 mm)
- **Z-Axis** 14" (356 mm)

6.2 TOOLING REFERENCE

6.2.1 Tooling Styles

The following tooling styles are available for the machine: quick-change tool post, turret, and gang tooling. You can mix all three options – for example, a gang plate mounted tool could be used in conjunction with tooling mounted in a turret.

Quick Change Tool Post Kit

Mount a manual Quick Change Tool Post Kit for 15L Slant-PRO Lathe (PN 33272) to the carriage for fast, repeatable manual tool changes.

The Quick Change Tool Post Kit:

- Uses a dovetail and tapered gib to hold tool holders to the post at adjustable, repeatable distances from the lathe bed so that individual tools can be brought up to centerline height of the spindle.
- Uses tool holders including standard 3/4 in. OD, ID (boring), and parting tool holders.

8-position Turret

The 8 Position Automatic Turret Kit for 15LX Lathe (PN 54578) allows automatic tool changing of both turning and boring tools as well as through-turret flood coolant.

With a Turret:

- **Tools Numbered 1-8** The machine automatically changes tools.
- **Tools Numbered 9-500** You can use gang tooling or quick change tooling on the lower portion of the carriage to change tools.

Gang Tooling

Mount a gang tooling plate (with three T-slots and a riser block) to the carriage to mount tool holders in industry-standard, gang-tool holding blocks.



Note: While gang tooling offers higher precision and faster tool change times than a manual tool post or turret, it requires vigilance to make sure that there's appropriate clearance between tool, workpiece, and chuck/spindle nose.

Gang tooling:

- Uses risers for both 1/2 in. and 3/4 in. centerline heights.
- Is ideal for small diameter, short parts.

- Can be swapped out as a single unit to reduce part setup time.

6.2.2 Tool Post Locations

Tools are mounted on the carriage in either a front tool post or rear tool post position:

- Front tool post tools move in +X direction.
- Rear tool post tools move in –X direction.

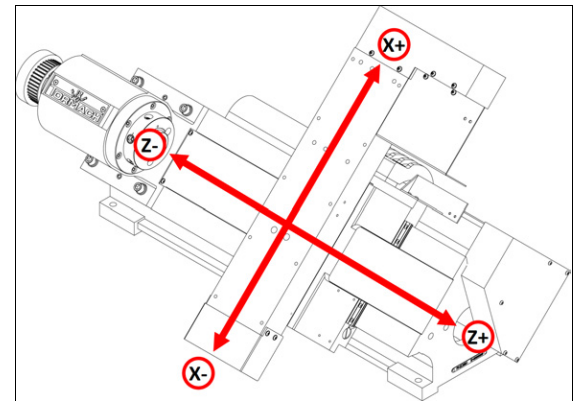


Figure 6-2: Axes directions on the machine.

The machine's three tooling configurations have different tool post locations, detailed in the following table:

Tooling Configuration	Tool Post Locations
Quick change tool post	Front tool post
Gang tooling	Front or rear tool post
Turret	Rear tool post

6.2.3 Tool Orientation

Tools have either a right-handed or left-handed orientation. You can mount right- or left-handed tools on the carriage in either the front tool post or rear tool post position. You can mount the tool either with the cutter facing toward you or turned with the cutter facing away from you.



Tip! Right-handed tools are commonly used for general purpose turning. Left-handed tools are sometimes used for back chamfering or grooving.

6: SYSTEM BASICS

6.2 Tooling Reference

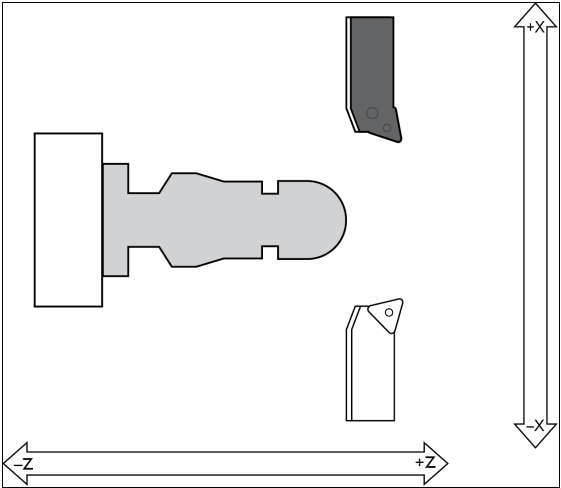


Figure 6-3: Left-handed tool orientation.

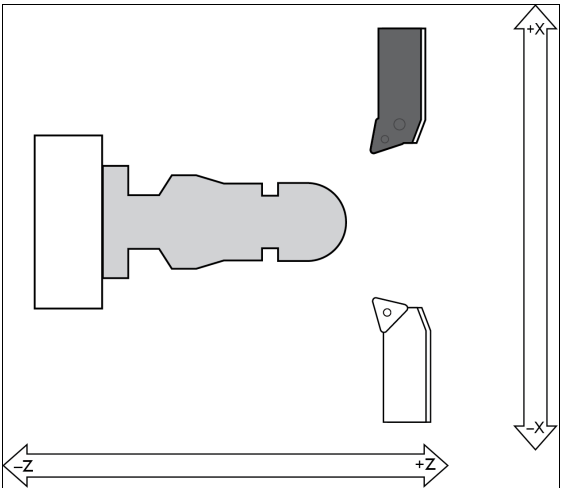


Figure 6-4: Right-handed tool orientation.

6.3 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.

6.3.1 Machine Controls

The following controls energize the machine's control electronics:

- The Main Disconnect switch, located on the front 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.

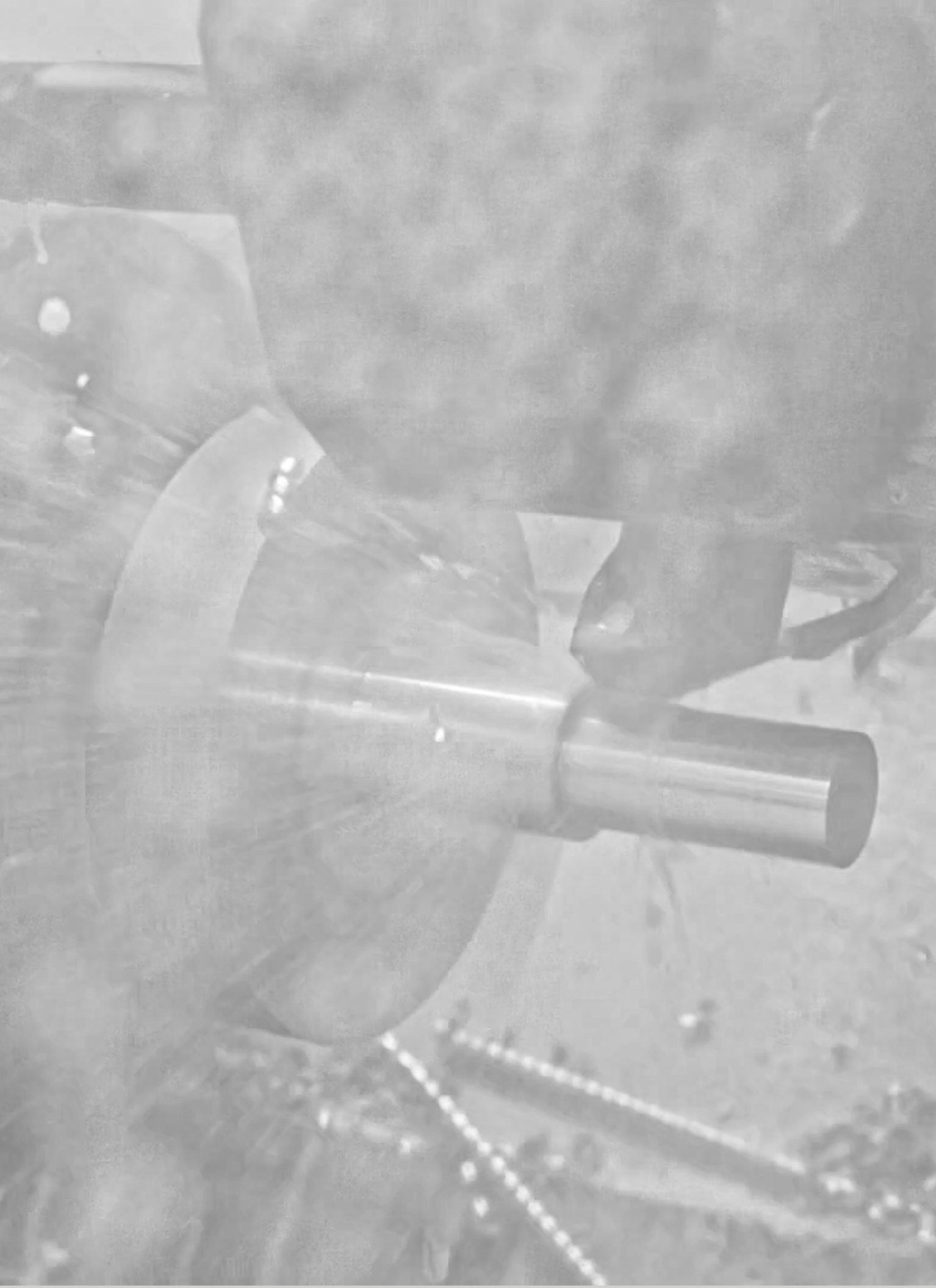
6.3.2 PathPilot Interface

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	72
7.2 Notebook Section	73
7.3 Persistent Controls	76
7.4 Keyboard Shortcuts	78
7.5 Manage PathPilot Versions	79

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.

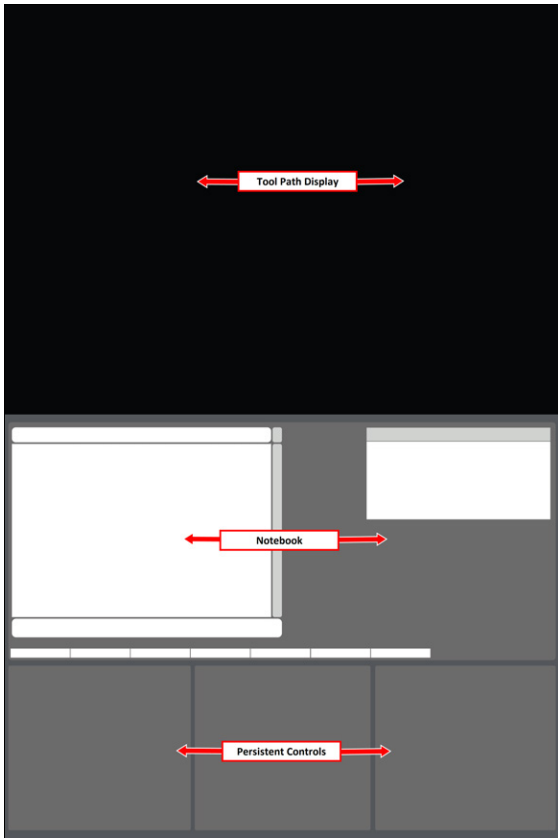


Figure 7-1: Sections in the PathPilot interface.

7.2 NOTEBOOK SECTION

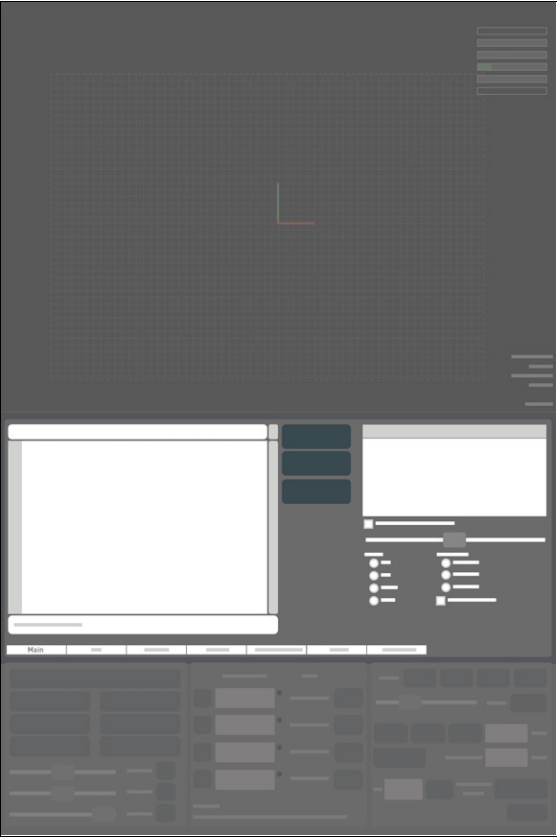


Figure 7-2: Notebook section.

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	73
7.2.2 File Tab	73
7.2.3 Settings Tab	74
7.2.4 Offsets Tab	74
7.2.5 Conversational Tab	74
7.2.6 Status Tab	74

7.2.1 Main Tab

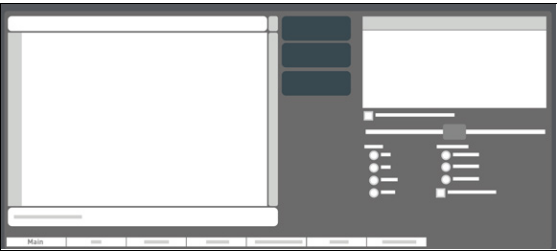


Figure 7-3: Main tab.

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 83); "Close the Current Program" (page 83).
- Send G-code commands directly to the machine using the Manual Data Input (MDI) Line DRO field.
For information, see "Manually Enter Commands" (page 123).
- 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 85); "Expand the G-Code Tab" (page 84); "Change the View of the Tool Path Display" (page 86).
- Make and restore backup files of your settings.
For information, see "Create Backup Files" (page 129); "Restore Backup Files" (page 130).

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 140).
- Edit G-code files.
For information, see "Edit G-Code" (page 83).
- Load .nc files into PathPilot to run a program.
For information, see "Load G-Code" (page 140).
- Move files within the system.
For information, see "Preview G-Code Files" (page 82); "Manage System Files" (page 129).

7: PATHPILOT INTERFACE OVERVIEW

7.2 Notebook Section

7.2.3 Settings Tab

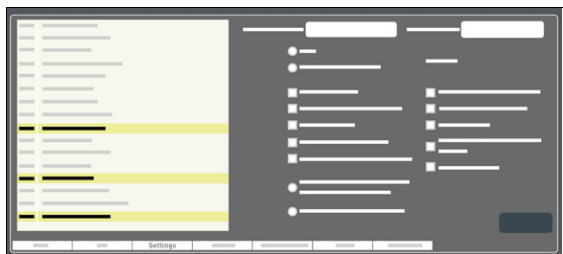


Figure 7-5: Settings tab.

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 103).
- Change the screen's layout orientation (landscape or portrait).
For information, see "Change the Screen Orientation" (page 109).
- Configure PathPilot for the accessories you're using.
For information, see "Enable the On-Screen Keyboard" (page 105); "Enable the USB M-Code I/O Interface Kit" (page 106); "Use a USB Camera" (page 106).
- Turn on feeds and speeds suggestions when using conversational programming.
For information, see "Enable Feeds and Speeds Suggestions in Conversational Routines" (page 125).
- Specify the way in which you want to use a G30 move.
For information, see "Limit G30 Moves" (page 104).
- Identify the available G-code modes that you can use.
For information, see "View Available G-Code Modes" (page 112).

7.2.4 Offsets Tab

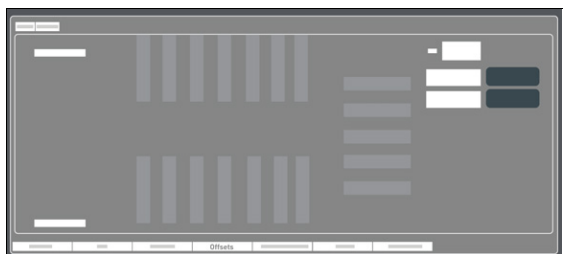


Figure 7-6: Offsets tab.

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 144).
- Work with a table of tool descriptions and tool offsets.
For information, see "Set Tool Geometry Offsets" (page 142).
- Read the currently programmed work offsets.
For information, see "View Work Offsets" (page 112).
- Change the tool number.
For information, see "Change the Tool Number" (page 122).

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 87).

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 103).

- Update or install a previous version of PathPilot.
For information, see "Manage PathPilot Versions"
(page 79).

7: PATHPILOT INTERFACE OVERVIEW

7.3 Persistent Controls

7.3 PERSISTENT CONTROLS

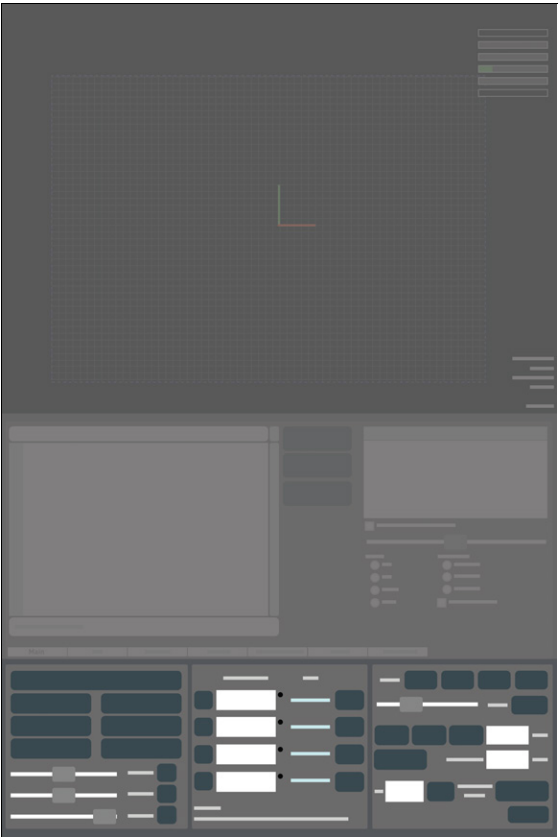


Figure 7-9: Persistent Controls section.

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.....	76
7.3.2 Position Status Area.....	76
7.3.3 Manual Control Area.....	77

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 135).
- Start, stop, or pause a G-code program.
For information, see "Start a Program" (page 115); "Stop Machine Motion" (page 115); "Use the Feed Hold Function" (page 117).
- Use overrides to change the feed rate, spindle speed, and maximum velocity.
For information, see "Use the Feed Rate Override Function" (page 117); "Use the Maxvel Override Function" (page 118); "Use the Spindle Override Function" (page 118).
- Manually control a G-code program.
For information, see "Use M01 Break Mode" (page 118); "Use Single Block Mode" (page 118).

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 141).
- Understand how you're jogging the machine.
For information, see "View the Active Axis to Jog" (page 113); "View the Current Machine Position" (page 114); "View the Distance to Go" (page 116).
- Quickly determine which G-code modes are active.
For information, see "View the Active G-Code Modes" (page 116).
- Change the spindle speed or feed rate.
For information, see "Change the Feed Rate Mode" (page 120); "Change the Spindle Speed Command Mode" (page 121).

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 137).
- Preset or go to a G30 position.
For information, see "Use a G30 Position" (page 122).
- Reference the machine axes.
For information, see "Reference the Machine" (page 136).
- Manually start and stop the spindle.
For information, see "About Spindle Controls" (page 121).
- Clamp and unclamp an automatic collet closer.

7: PATHPILOT INTERFACE OVERVIEW

7.4 Keyboard Shortcuts

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

7.5 MANAGE PATHPILOT VERSIONS



Note: To update PathPilot on a CE machine, you must first confirm that the machine is in setup mode.

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**.

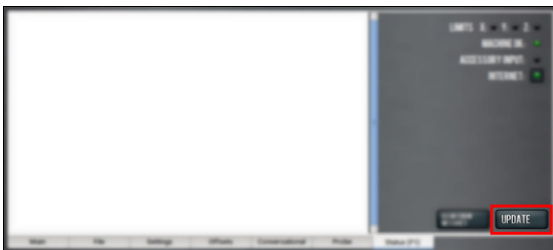


Figure 7-13: Update button on the Status tab.

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

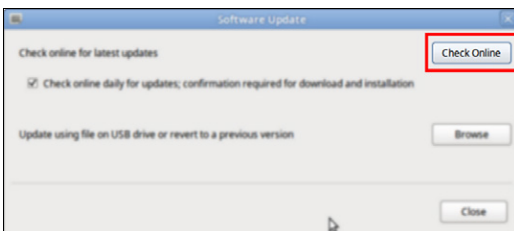


Figure 7-14: Software Update dialog box.

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**.

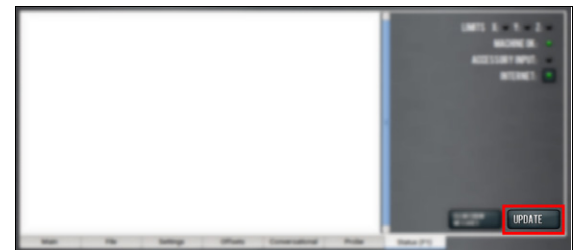


Figure 7-16: Update button on the Status tab.

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

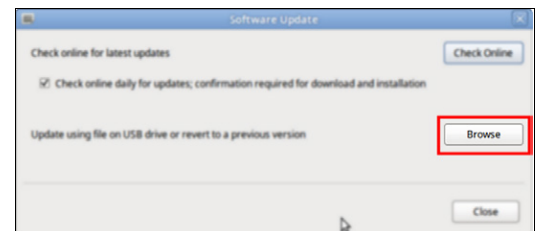


Figure 7-17: Software Update dialog box.

7: PATHPILOT INTERFACE OVERVIEW

7.5 Manage PathPilot Versions

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



Figure 7-18: Browse dialog box.

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

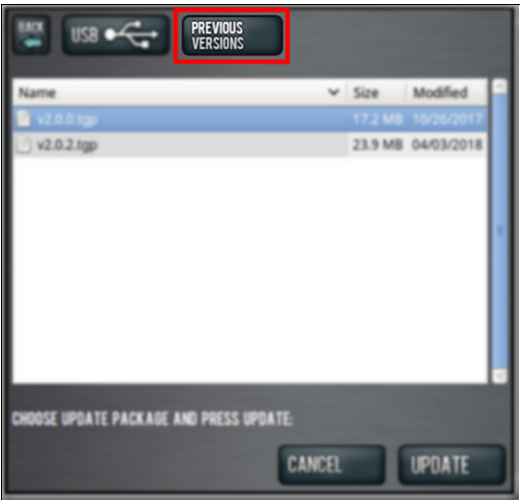


Figure 7-21: Browse dialog box.

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

7.5.3 Install a Previous Version of an Update File

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

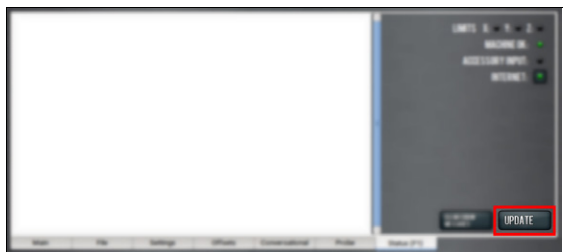


Figure 7-19: Update button on the Status tab.

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

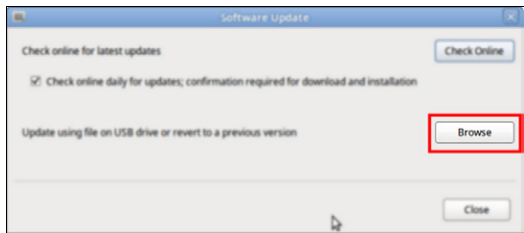


Figure 7-20: Software Update dialog box.

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

PATHPILOT TOOLS AND FEATURES

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

8.1 Create and Load G-Code Files	82
8.2 Machine Settings and Accessories	103
8.3 Set Up G-Code Programs	110
8.4 Run G-Code Programs	113
8.5 Control G-Code Programs	117
8.6 System File Management	129

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	82
8.1.2 Edit G-Code	83
8.1.3 Read G-Code	84
8.1.4 Use Conversational Programming	87

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 140).

To load G-code:

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

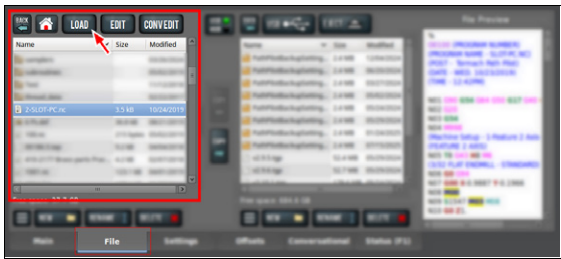


Figure 8-1: Controller Files window on the File tab.

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).

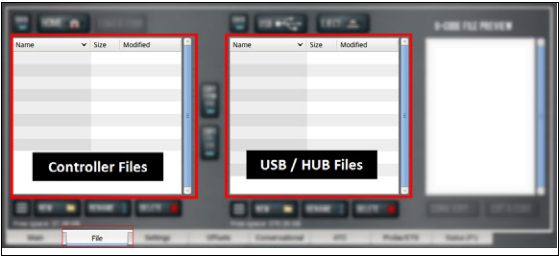


Figure 8-2: File tab.

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 →**.

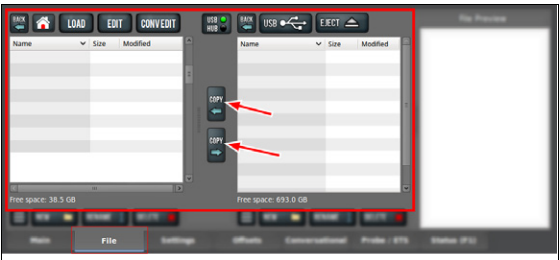


Figure 8-3: File tab.

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.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

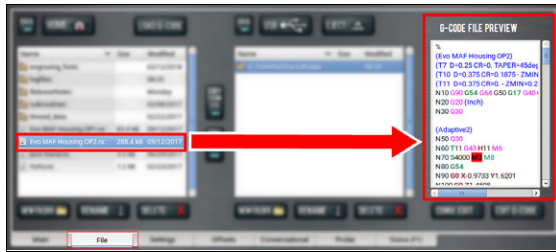


Figure 8-4: File tab.

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.

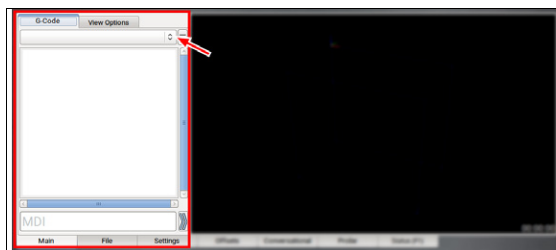


Figure 8-5: Recent Files menu on the Main tab.

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** 83
- **Edit G-Code with Conversational Programming** 83

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 140).

To edit G-code with a text editor:

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

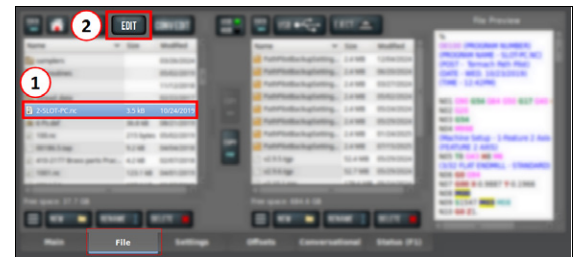


Figure 8-7: Edit G-code button on the File tab.

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 140).

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.

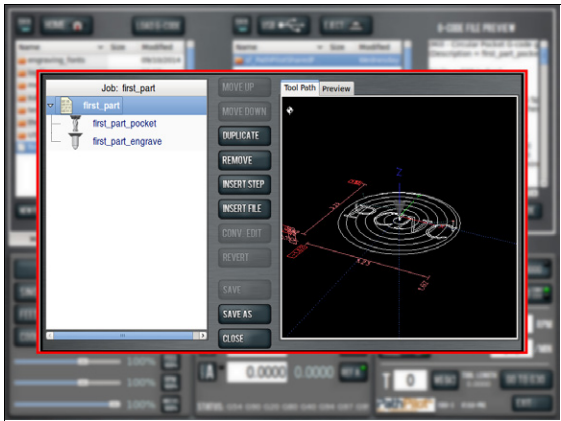


Figure 8-8: Job assignment editor.

- 3. Edit the file contents as needed. Do any of the following:
- | | |
|-----------------------------------|----|
| Change the Step Order..... | 84 |
| Create a New Job Assignment..... | 84 |
| Load an Existing G-Code File..... | 84 |
| Edit a Job Assignment..... | 84 |
- 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.....	84
Search in the Code.....	85
Set a New Start Line.....	85
Change the View of the Tool Path Display.....	86

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**.

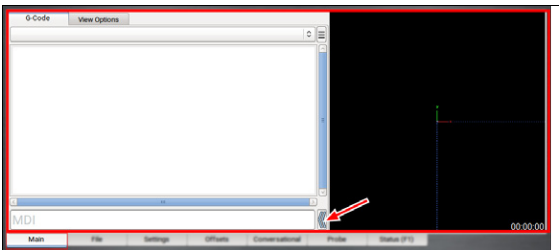


Figure 8-9: Window Expander on the Main tab.

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.

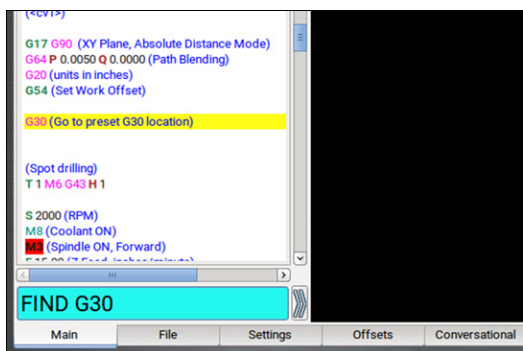


Figure 8-10: Search for a text command.

- **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.

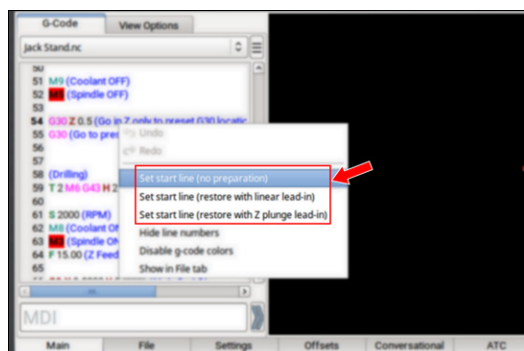


Figure 8-12: Accessing the Options menu by right-clicking.

- 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

- 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.

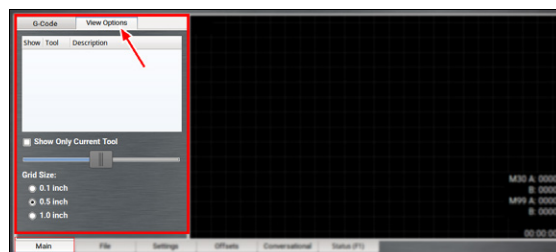


Figure 8-14: View Options tab on the Main tab.

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.....	87
Create an Outside Diameter.....	87
Create an Internal Diameter.....	89
Create a Profile on a Part.....	92
Create a Face on a Part.....	94
Create a Chamfer or Radius on a Part.....	96
Create a Groove or Part a Workpiece.....	97
Create Holes on a Part or Tap a Hole.....	99
Create Threads on a Part.....	100

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 89).

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 rear tool, and type 3 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.

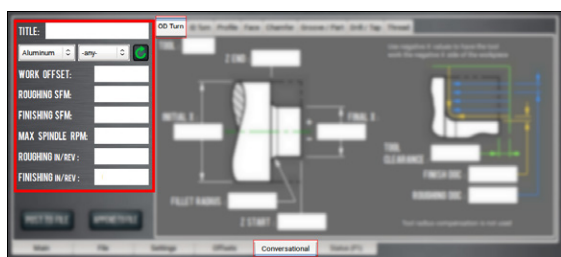


Figure 8-15: Conversational DROs on the OD Turn tab.

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 away from you). If they're both negative, the tool works the negative side of the spindle (the side closer to 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).

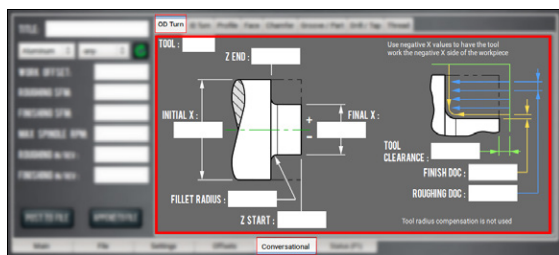


Figure 8-16: Program-specific DROs on the OD Turn Tab.

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 91).

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 2 for a front tool, and type 3 for a rear 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.

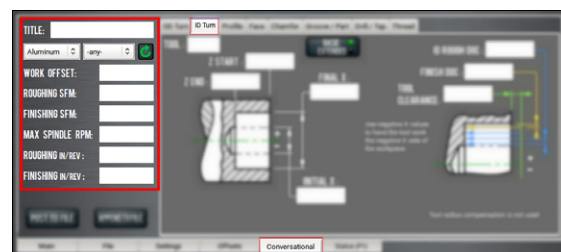


Figure 8-17: ID Turn tab on the Conversational tab.

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).

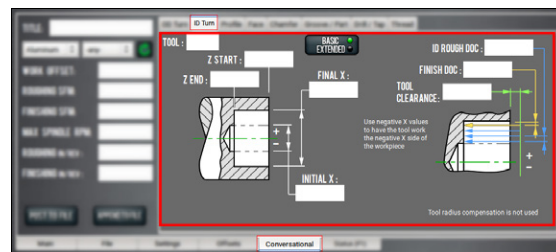


Figure 8-18: Program-specific DROs on the ID Turn tab.

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.

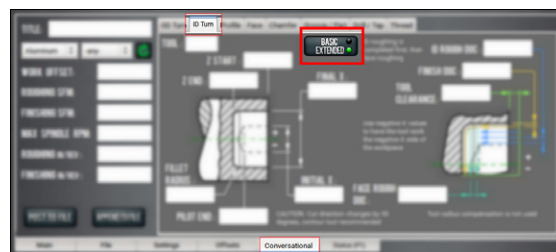


Figure 8-19: ID Turn tab in Extended mode.

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

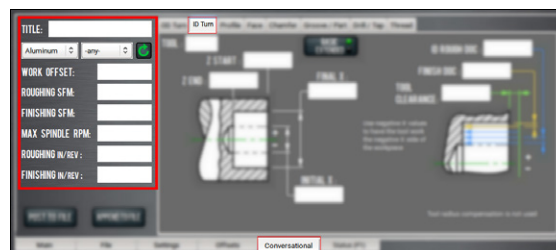


Figure 8-20: ID Turn tab on the Conversational tab.

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 ((INITIAL X - 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).

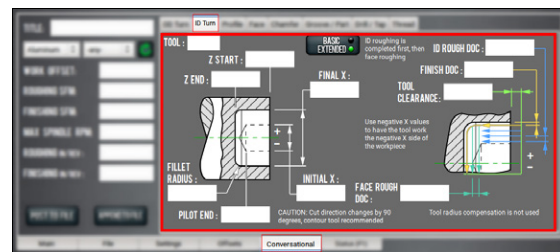


Figure 8-21: Program-specific DROs on the ID Turn tab.

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 End - [2 x 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 94).

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 smaller diameter than first X value on the Profile Point table.
- An internal tool is indicated in the Tool DRO field.
- Verify that the X-axis direction matches your specific lathe.

Complete the following steps in the order listed:

Describe the Stock 92
Identify the Profile Points 93
Identify the Tool 93
Identify the Roughing and Finishing 93
Describe the Tool Geometry 94
About Profiling 94

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.

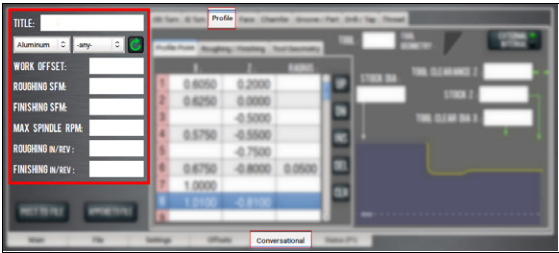


Figure 8-22: Conversational DROs on the Profile tab.

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 larger 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 smaller diameter than the first diameter (specified in the Profile Point table).

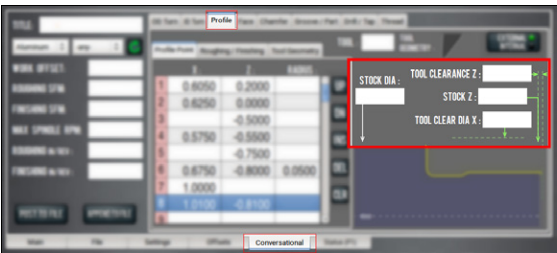


Figure 8-23: Program-specific DROs on the Profile tab.

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 15L lathes, the X+ values **increase** outward from the spindle. When using a quick-change tool post, you're touching off the front tool post with a negative X value. PathPilot displays these X values in the **Profile** tab as **positive** values.



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.

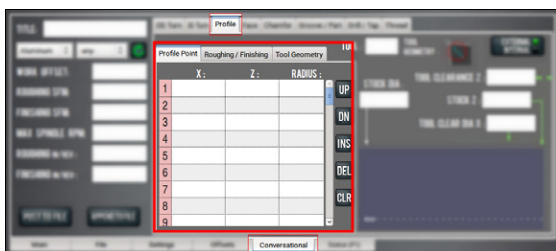


Figure 8-24: Profile Points table on the Profile tab.

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.

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.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

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).

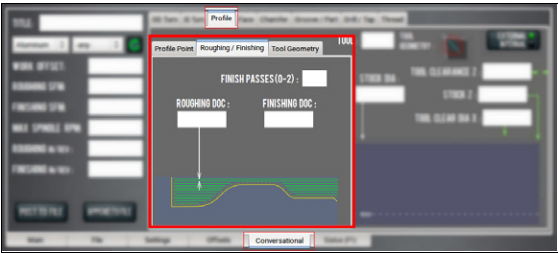


Figure 8-25: Roughing/Finishing tab on the Profile tab.

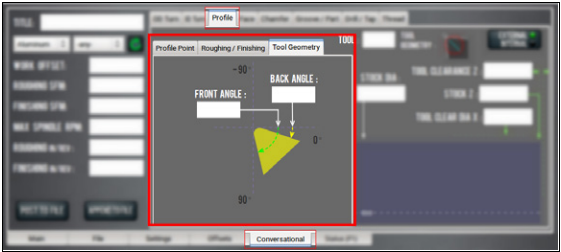


Figure 8-26: Tool Geometry tab on the Profile tab.



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°

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 15L lathes, an X+ cutting direction moves outward from 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°

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 away from 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 closest 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.

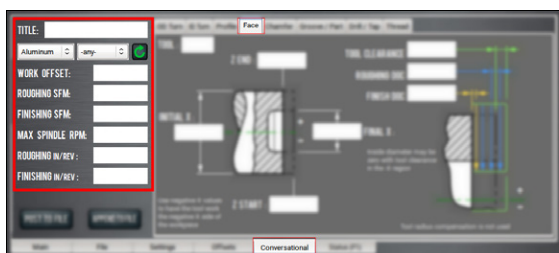


Figure 8-27: Conversational DROs on the Face tab.

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).

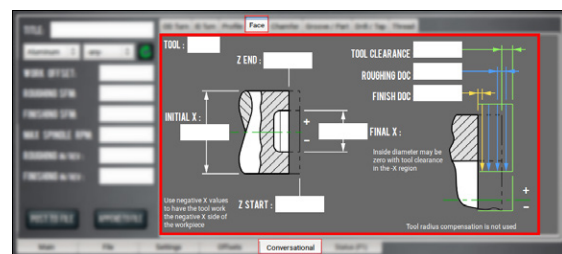


Figure 8-28: Program DROs on the Face tab.

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.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

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" (below).

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.

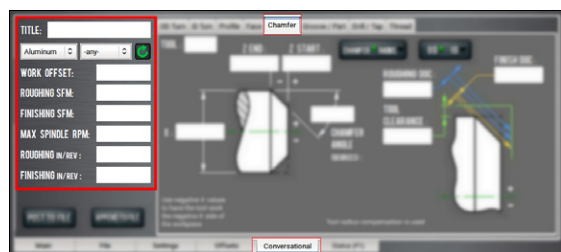


Figure 8-29: Conversational DROs on the Chamfer tab.

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).

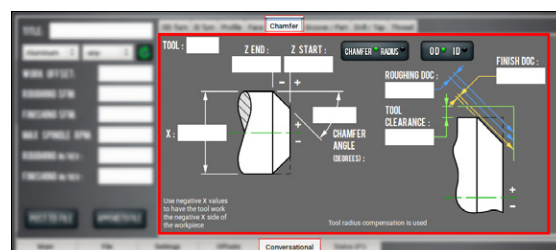


Figure 8-30: Program DROs on the Chamfer tab.

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" (on the next page).

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 negative side of Z Start, Back Tool = Type 1
 - Groove on the positive side of Z Start, Back Tool = Type 2
 - 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, Back Tool = Type 1
 - 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.

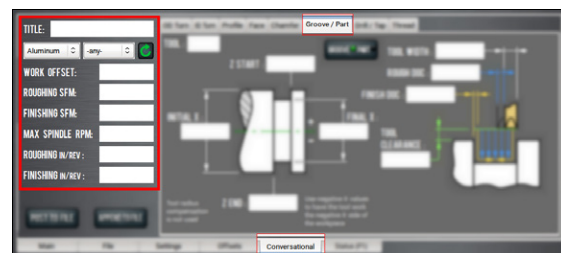


Figure 8-31: Conversational DROs on the Groove/Part tab.

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

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

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.

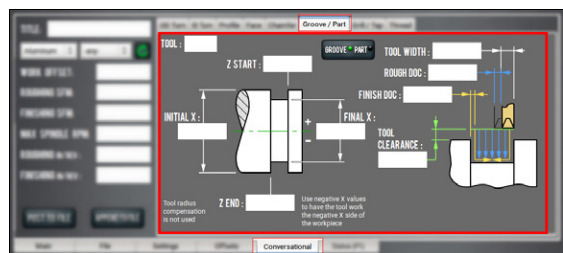


Figure 8-32: Program-specific DROs on the Groove/Part tab.

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.

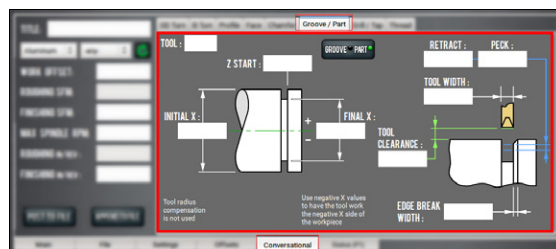


Figure 8-33: Program-specific DROs on the Groove/Part tab.

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 8-34: 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" (below).

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.
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.

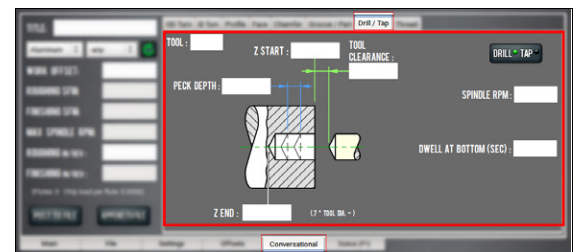


Figure 8-35: Program-specific DROs on the Drill/Tap tab.

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.

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

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.

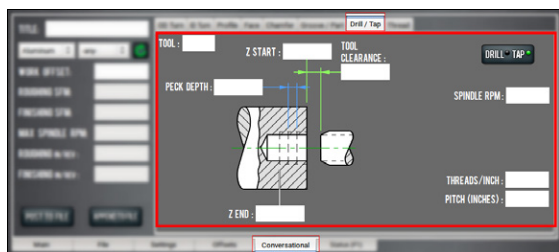


Figure 8-36: Program-specific DROs on the Drill/Tap tab.

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:

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

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

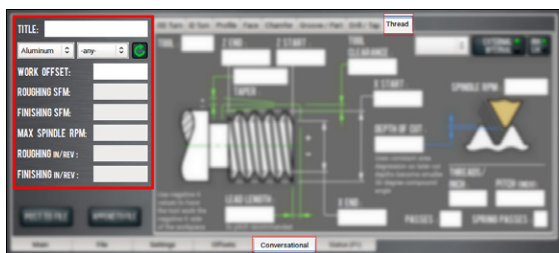


Figure 8-37: Conversational DROs on the Thread tab.

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



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.

- 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.

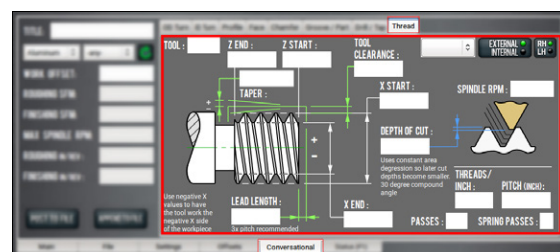


Figure 8-38: Program-specific DROs on the Thread tab.

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

8: PATHPILOT TOOLS AND FEATURES

8.1 Create and Load G-Code Files

material by cross sectional area. The first pass has the most X displacement, and this decreases with each pass.

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	103
8.2.2 Change the Network Name	103
8.2.3 Specify the Spindle Pulley Option	104
8.2.4 Specify the Tool Change Method	104
8.2.5 Limit G30 Moves	104
8.2.6 Enable the On-Screen Keyboard	105
8.2.7 Enable Tooltips	105
8.2.8 Enable the USB M-Code I/O Interface Kit	106
8.2.9 Enable Feeds and Speeds Suggestions in Conversational Routines	106
8.2.10 Use a USB Camera	106
8.2.11 Change the Screen Orientation	109

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 8-39: Internet button on the Settings tab.

The **Network Configuration** dialog box displays.

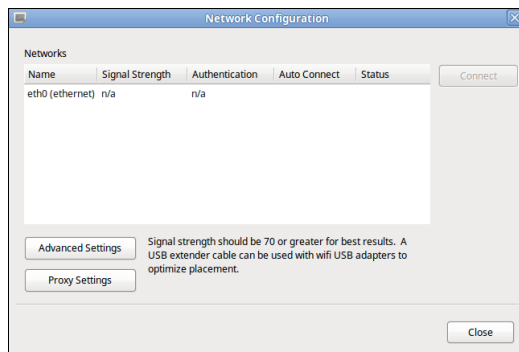


Figure 8-40: Network Configuration dialog box.

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.

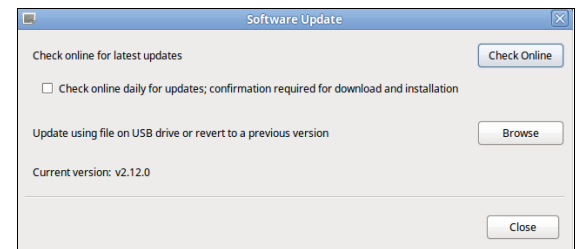


Figure 8-41: Software Update dialog box.

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.

8.2.2 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

8: PATHPILOT TOOLS AND FEATURES

8.2 Machine Settings and Accessories

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.

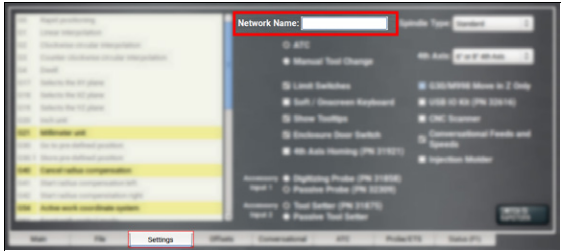


Figure 8-42: Network Name field on the Settings tab.



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 Specify the Spindle Pulley Option

- From the Settings tab, select one of the following:
 - **3500 RPM (Collet)**
 - **2500 RPM (3-Jaw Chuck)**
 - **2000 RPM (4-Jaw Chuck)**

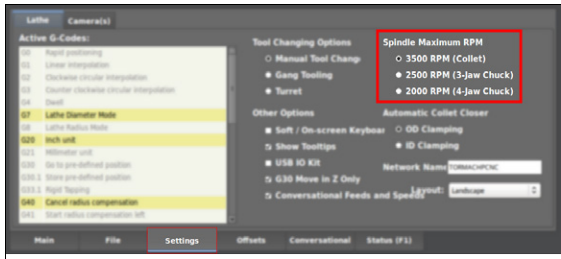


Figure 8-43: Spindle pulley options on the Settings tab.

8.2.4 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.

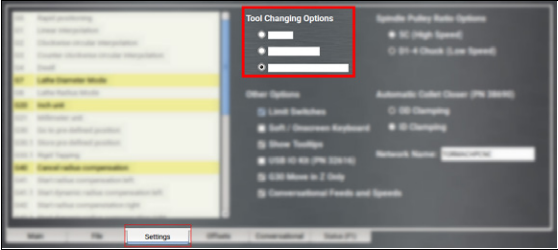


Figure 8-44: Tool change method options on the Settings tab.

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.

8.2.5 Limit G30 Moves

You can limit G30 moves so that only the Z-axis moves. For information, see "About G30" (page 123).

To limit G30 moves:

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

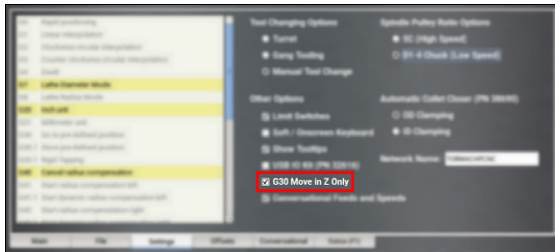


Figure 8-45: Settings tab.

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 122).

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" (on the previous page).



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.

8.2.6 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**.

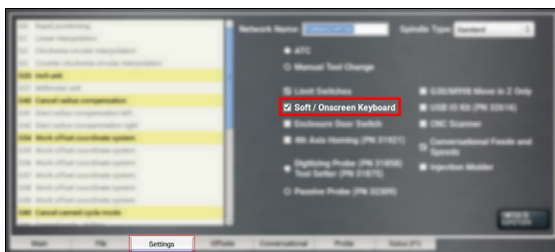


Figure 8-46: Settings tab.

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 8-47: Soft (on-screen) keyboard.

8.2.7 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**.

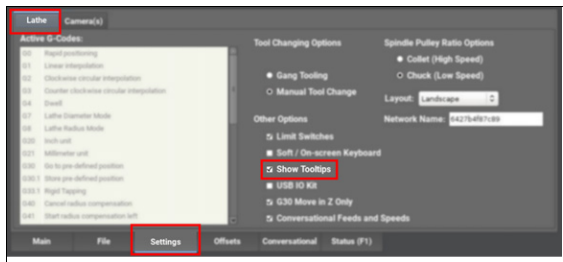


Figure 8-48: Show Tooltips checkbox.



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.

8.2.8 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)**.

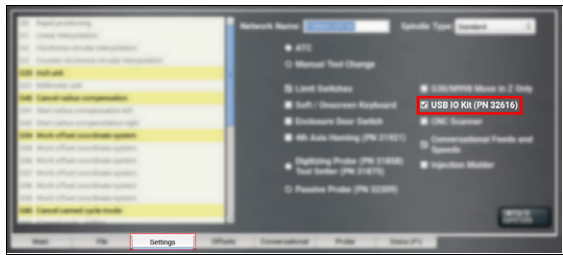


Figure 8-49: Settings tab.

8.2.9 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 124).

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

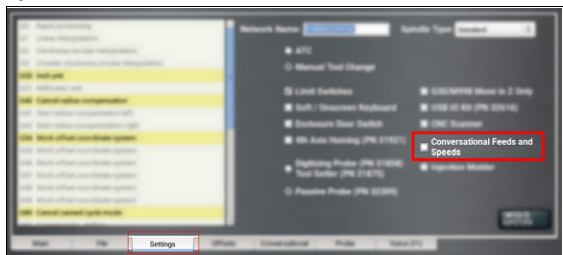


Figure 8-50: Settings tab.

8.2.10 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 8-51: 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 8-52: Manual recording controls.

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

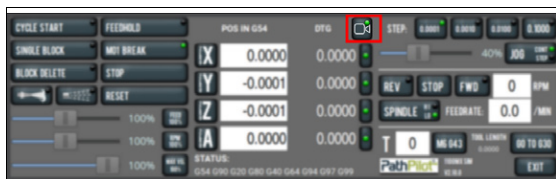


Figure 8-53: Video Camera Recording button.

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 8-54: 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.

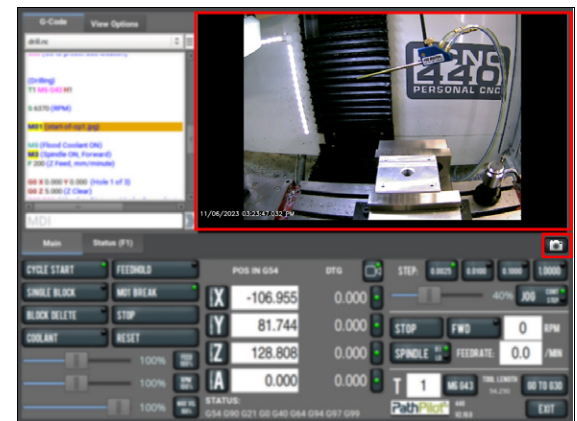


Figure 8-55: Example of taking a photo.

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 167).

8.2.11 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.

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.

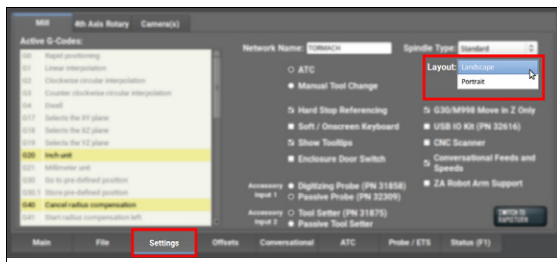


Figure 8-56: Layout drop-down menu on the Settings tab.

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.

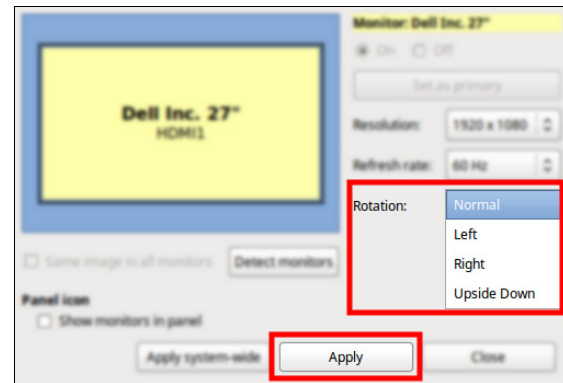


Figure 8-57: Monitor configuration dialog box.

The controller restarts in portrait layout.

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. 110
8.3.2 Touch Off the Tool Geometry Offsets. 110
8.3.3 Set Work Offsets. 111
8.3.4 View Work Offsets. 112
8.3.5 View Available G-Code Modes. 112

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 143).

Note: You can import a .csv file with tool geometry offset data. For information, see "Import and Export the Tool Table" (page 144).

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.

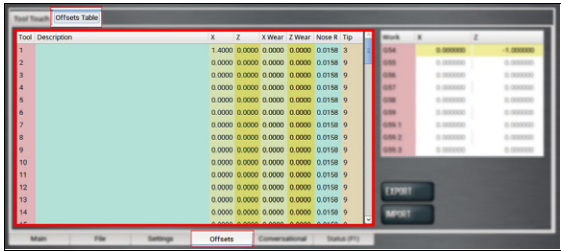


Figure 8-58: Tool Table window on the Offsets tab.

- 4. Touch off the tool geometry offsets. For information, see "Touch Off the Tool Geometry Offsets" (page 143).
- 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).
The tools along top are rear tool post tools (used by machines with a turret 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 negative.

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 negative 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 negative X (diameter) values, and most X words in part programming for these tools have negative 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**.

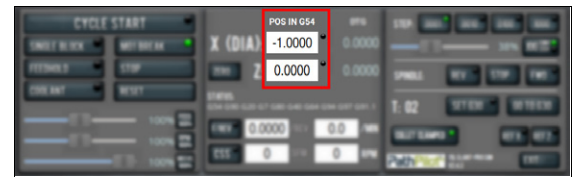


Figure 8-59: Work Offset DRO fields.

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 8-60: 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 141).

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

8: PATHPILOT TOOLS AND FEATURES

8.3 Set Up G-Code Programs

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

8.3.4 View Work Offsets

- To view the current work offset:
- From the **Offsets** tab, on the **Offsets Table** tab, identify the **Work Offsets Table** window.

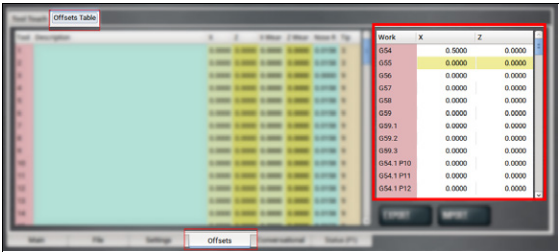


Figure 8-61: Work Offsets Table window.

The active work offset is highlighted.

To change the current work offset, go to "Set Work Offsets" (page 141).

8.3.5 View Available G-Code Modes

The G-Code Description window shows a list of all available G-code modes.

To view available G-code modes:

- From the **Settings** tab, find the **G-Code Description** window.

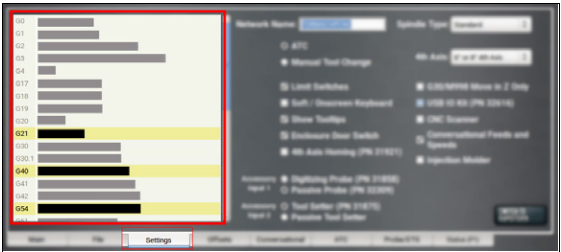


Figure 8-62: G-code Description window on the Settings tab.

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	113
8.4.2 View the Active Axis to Jog	113
8.4.3 Jog the Machine	113
8.4.4 View the Current Machine Position	114
8.4.5 Reference the Machine	115
8.4.6 Start a Program	115
8.4.7 Stop Machine Motion	115
8.4.8 Operate the Coolant Pump	115
8.4.9 View the Active G-Code Modes	116
8.4.10 View the Distance to Go	116

8.4.1 Bring the Machine Out of Reset

- Select **Reset**.

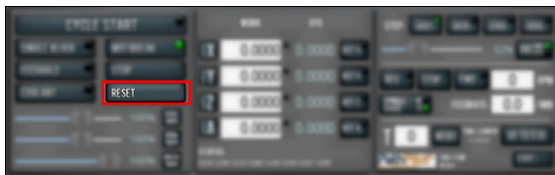


Figure 8-63: Reset button.

For more information on reset mode, see "About Reset Mode" (page 135).

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.

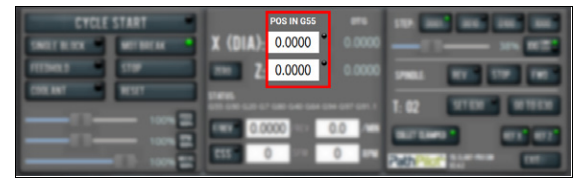


Figure 8-64: Work Offset DRO fields.

For information, see "Jog the Machine" (page 137).

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.

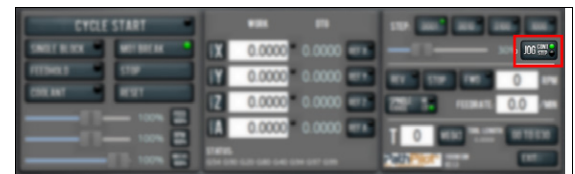


Figure 8-65: Jog button.

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.

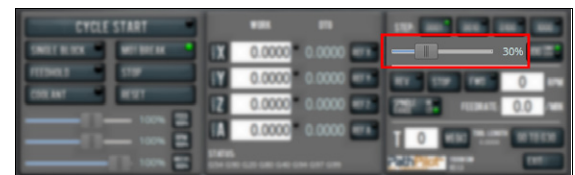


Figure 8-66: Jog Speed slider.

For more information on continuous velocity mode, see "About Continuous Velocity Jogging" (page 137).

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 8-67: 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 137).

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**.

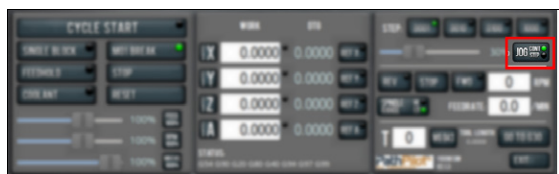


Figure 8-68: Continuous velocity jogging controls.

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.

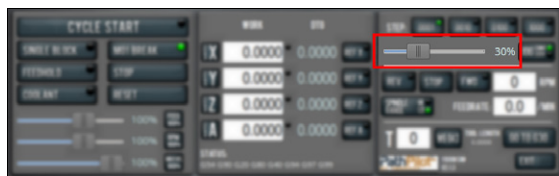


Figure 8-69: Jog Speed slider.

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.

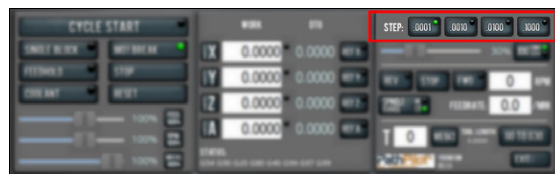


Figure 8-70: Step buttons (in G20 mode).

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.

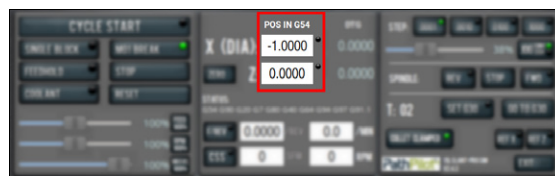


Figure 8-71: Work Offset DRO fields.

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 141).

8.4.5 Reference the Machine



Note: The machine automatically references when it's powered on.

1. Verify that the machine can freely move to its reference position (at the ends of travel).
2. To verify that the tooling is clear of any possible obstructions, reference the Z-axis before referencing the other axes: from the PathPilot interface, select **Ref Z**.



Figure 8-72: 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.

About Referencing

The axes are referenced during assembly and do not need re-referencing after powering on or once the Emergency Stop button is released. You must reference the X- or Z-axis at the following times:

- After replacement of a linear motion component such as linear rails, ballscrew, or servo motor
- Disconnection of an encoder battery and powering off of the machine
- Disconnection of the encoder cable from the servo motor

When referencing the X- and Z-axis, the machine moves to the end of travel, presses against the bumper on the ballscrew until torque increases, and then backs off to the index pulse of the encoder.

8.4.6 Start a Program

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

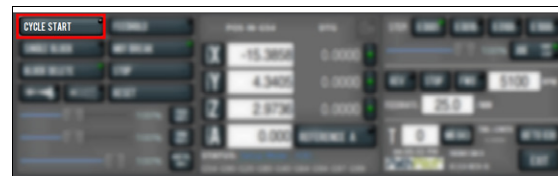


Figure 8-73: Cycle Start button.

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 73).
- Before you've loaded a G-code program. For information, see "Load G-Code" (page 140).
- Before referencing the machine. For information, see "Reference the Machine" (page 136).

8.4.7 Stop Machine Motion

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

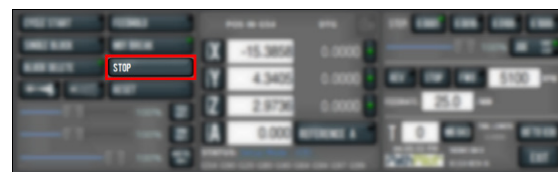


Figure 8-74: Stop button.

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 8-75: Coolant button.

For more information on turning on and off coolant, see "About Coolant" (page 145).

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 8-76: Status read-only DRO field.

For more information on G-code modes, go to "View Available G-Code Modes" (page 112).

8.4.10 View the Distance to Go

To view the distance to go:

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

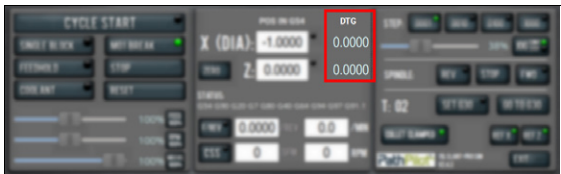


Figure 8-77: DTG read-only DRO fields.

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	117
8.5.2 Use the Feed Rate Override Function	117
8.5.3 Use M01 Break Mode	118
8.5.4 Use the Maxvel Override Function	118
8.5.5 Use Single Block Mode	118
8.5.6 Use the Spindle Override Function	118
8.5.7 Change the Feed Rate	119
8.5.8 Change the Spindle Speed	119
8.5.9 Change the Feed Rate Mode	120
8.5.10 Change the Spindle Speed Command Mode	121
8.5.11 Change the Tool Number	122
8.5.12 Use a G30 Position	122
8.5.13 Manually Enter Commands	123
8.5.14 Copy Recently Entered Commands	124
8.5.15 Use Feeds and Speeds Suggestions	124
8.5.16 Use Cycle Counters (M30 and M99)	127

8.5.1 Use the Feed Hold Function

- Select **Feed Hold**.

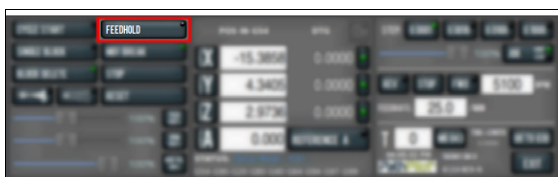


Figure 8-78: Feed Hold button.



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 115).



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.

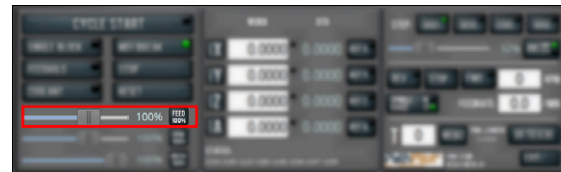


Figure 8-79: Feed Rate Override slider.



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**.

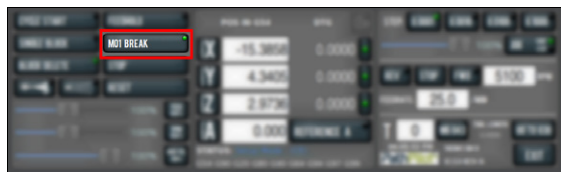


Figure 8-80: M01 Break button.

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 115).
- **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 8-81: 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**.

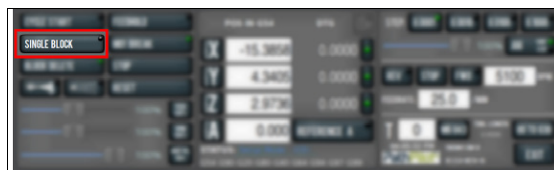


Figure 8-82: Single Block button.

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 115).

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 8-83: 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 8-84: 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 8-85: Spindle RPM DRO field.

For information, see "About Spindle Controls" (page 121).

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 8-86: 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 8-87: 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.

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.

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.

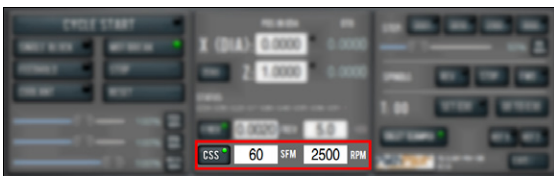


Figure 8-88: SFM DRO field.

To turn on constant RPM mode (G97):

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

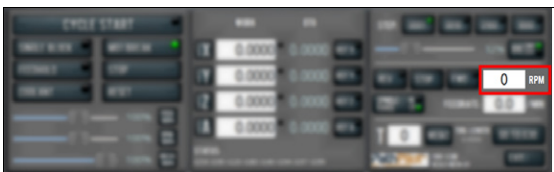


Figure 8-89: RPM DRO field.

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.

8.5.11 Change the Tool Number

The Tool DRO field shows the tool number currently active.



Figure 8-90: 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 8-91: 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 8-92: Set G30 button.

To go to a set G30 position:

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



Figure 8-93: 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 104).



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.

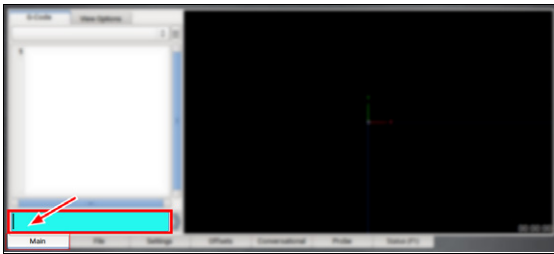


Figure 8-94: MDI Line DRO field.

The DRO field highlights.

2. Type the command.



Note: You can use the **Backspace**, **Delete**, **Left Arrow**, and **Right Arrow** keys to correct typing errors.

3. You must press the **Enter** key to execute the command.
To abandon the command, press **Esc**.

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.

8.5.14 Copy Recently Entered Commands

- From the **MDI Line** DRO field, press either the **Up Arrow** key or the **Down Arrow** key.
The previously entered command displays.
- 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).

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.

- If you haven't yet done so, enable the conversational feeds and speeds setting. From the **Settings** tab, select **Conversational Feeds and Speeds**.

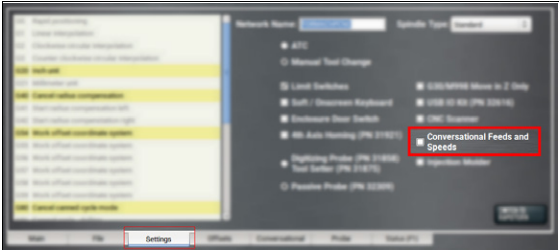


Figure 8-95: Settings tab.

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



Figure 8-96: Feeds and speeds suggestions on the Conversational tab.

- From the **Material** dropdown, select your material (like **Aluminum** or **Plastic**).
- If required, from the **Sub-Type** dropdown, select the material sub-type (like **-any-** or **6061**).
- In the **Tool** DRO field, type the assigned tool number.

6. Select **Refresh** (to the right of the **Sub-Type** dropdown).
The following machining-related DRO fields are calculated:
 - **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.

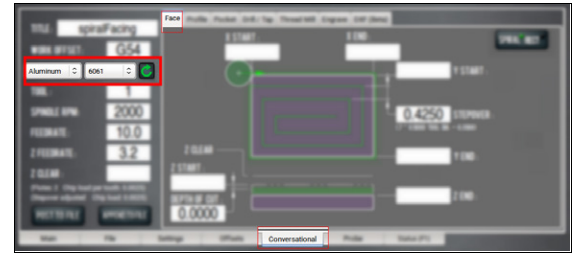


Figure 8-97: Refresh button on the Conversational tab.

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**.

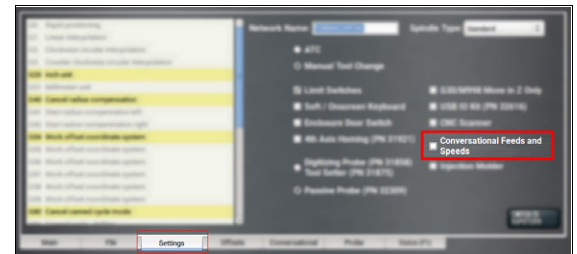


Figure 8-98: Settings tab.

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.

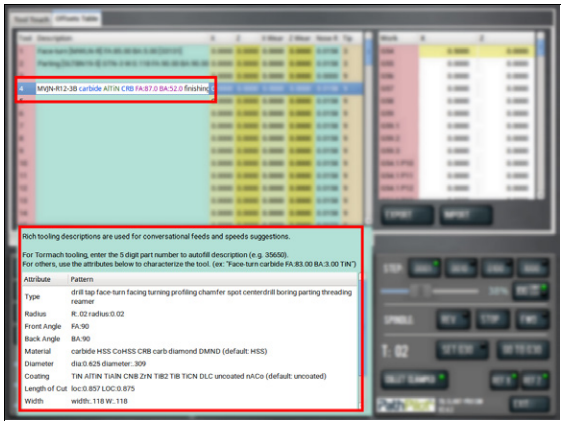


Figure 8-99: A manually-entered tool description.

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.

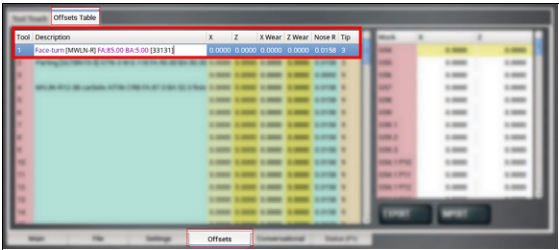


Figure 8-100: An automatically-generated tool description for a Tormach tool.

- 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.

Table with 4 columns: Item, Pattern, Example, Notes. It lists tool types and their corresponding patterns and examples for recognition by PathPilot.

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	129
8.6.2 Create Backup Files	129
8.6.3 Restore Backup Files	130
8.6.4 Import and Export the Tool Table	131

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 140).

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. Do not use the PathPilot v2.0 Upgrade.

2. From the PathPilot interface, on the **Main** tab, in the **MDI Line DRO** field, type `ADMIN SETTINGS BACKUP`. Then select the **Enter** key. PathPilot generates a backup .zip file, and the **Admin Settings Backup** dialog box displays.

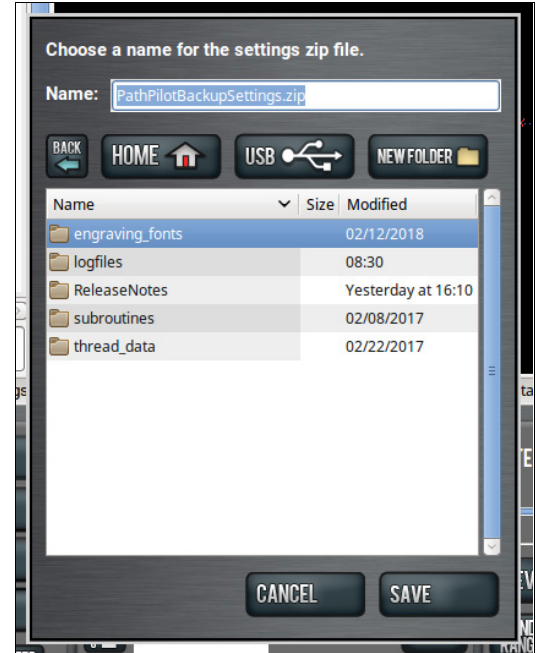


Figure 8-101: Admin Settings Backup dialog box.

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.

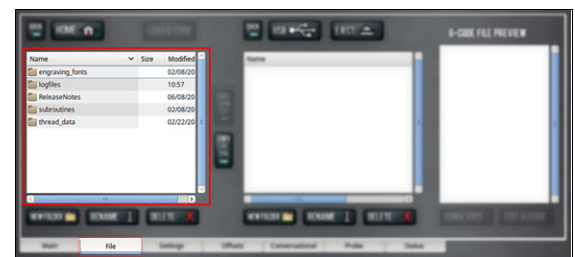


Figure 8-102: Controller Files window on the File tab.

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.
- Go to Install PathPilot v2.x.

About Backup Files

Make a regular backup of all tool offset and fixture information and machine settings stored on your PathPilot controller. Store the file externally to use if you replace your controller or restore it to factory settings.

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.

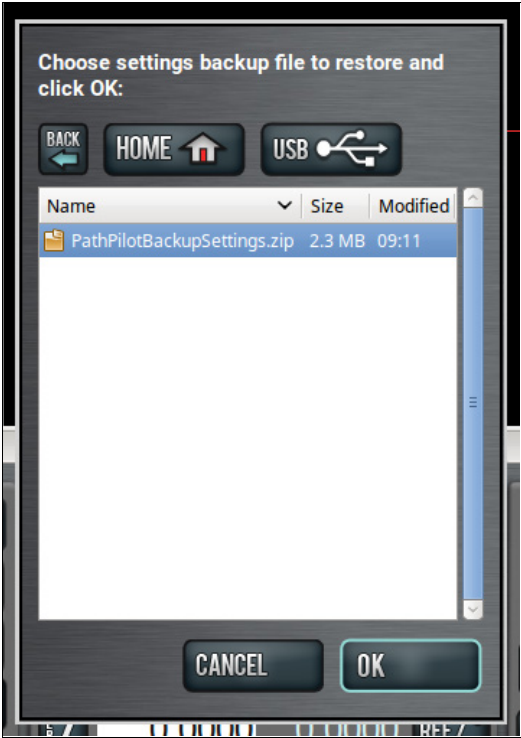


Figure 8-103: Admin Settings Restore dialog box.

- 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.

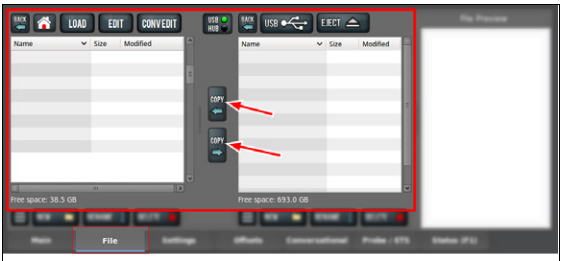


Figure 8-104: USB Files window on the File tab.

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.

7. If desired, you can enable an Internet connection in the PathPilot controller, which allows you to use Dropbox with PathPilot, and receive automatic PathPilot updates. Go to the next section, "Enable an Internet Connection" (page 103).
If you don't want to use these features, you have completed installing and configuring PathPilot v2.x.

8.6.4 Import and Export the Tool Table

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

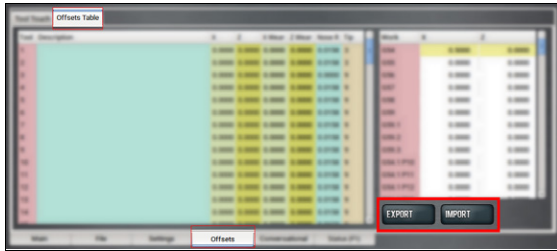


Figure 8-105: Export and Import buttons on the Offsets tab.

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.

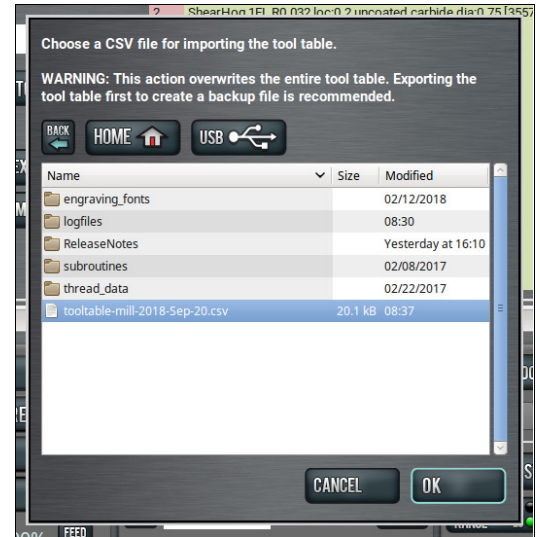


Figure 8-106: Import dialog box.

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.

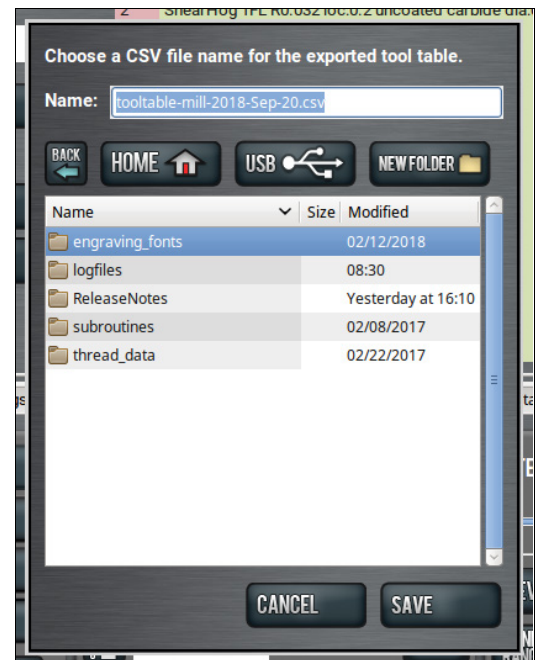


Figure 8-107: Export dialog box.

2. In the **Name** DRO field, type the name for the .csv file.

8: PATHPILOT TOOLS AND FEATURES

8.6 System File Management

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.....	134
9.2 Bring the Machine Out of Reset.....	135
9.3 Reference the Machine.....	136
9.4 Jog the Machine.....	137
9.5 Manually Control the Spindle.....	139
9.6 Load G-Code.....	140
9.7 Set Work Offsets.....	141
9.8 Set Tool Geometry Offsets.....	142
9.9 Touch Off the Tool Geometry Offsets.....	143
9.10 Operate the Coolant Pump.....	145

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**.

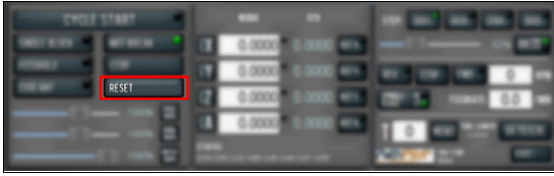


Figure 9-1: Reset button.

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



Note: The machine automatically references when it's powered on.

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.

9.3.1 About Referencing

The axes are referenced during assembly and do not need re-referencing after powering on or once the Emergency Stop button is released. You must reference the X- or Z-axis at the following times:

- After replacement of a linear motion component such as linear rails, ballscrew, or servo motor
- Disconnection of an encoder battery and powering off of the machine
- Disconnection of the encoder cable from the servo motor

When referencing the X- and Z-axis, the machine moves to the end of travel, presses against the bumper on the ballscrew until torque increases, and then backs off to the index pulse of the encoder.

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.

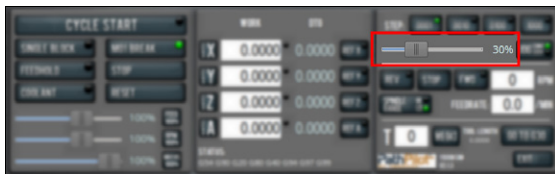


Figure 9-4: Jog Speed slider.

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.

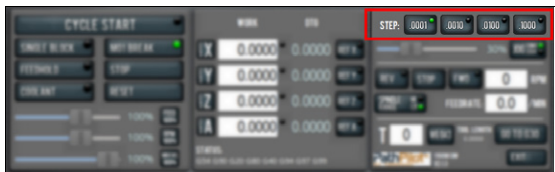


Figure 9-5: Step buttons (in G20 mode).

- 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

- **Axis Movement**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.



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

- **Axis Selection**Four buttons let you toggle between the X, Y, Z, and A axes.



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**A dedicated Step button toggles between available jog step sizes.

9.5 MANUALLY CONTROL THE SPINDLE

1. Verify that the machine is powered on and out of reset.
2. In the **RPM** DRO field, type the desired RPM speed. Then select the **Enter** key.

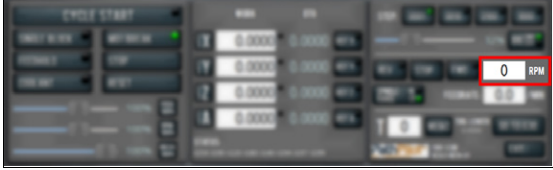


Figure 9-6: RPM DRO field.

3. Select **FWD** to start the spindle in the forward direction, or **REV** to start the spindle in the reverse direction.
4. Select **Stop** to stop the spindle.

9.5.1 About the Spindle

The machine spindle rotates the workpiece, which is then machined upon by a cutting tool to remove material.

Operate the spindle either manually or by G-code commands (entered in the MDI Line DRO field or programmed into a G-code program).

The machine's spindle rotates either clockwise (forward) or counterclockwise (reverse) at a specified spindle speed.

9: BASIC OPERATIONS

9.6 Load G-Code

9.6 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**.

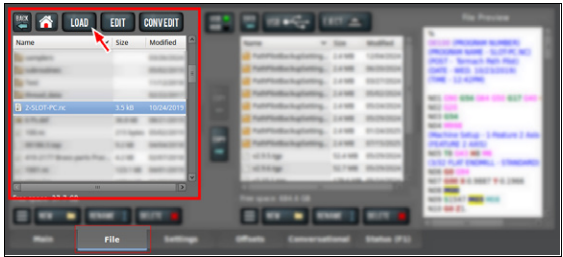


Figure 9-7: Controller Files window on the File tab.



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

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

9.6.1 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).

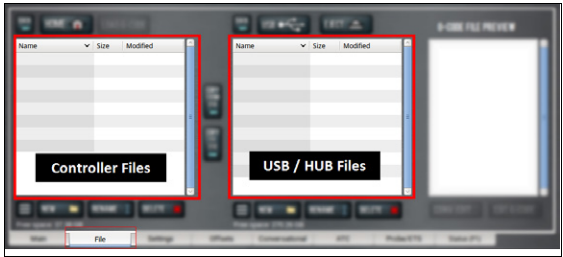


Figure 9-8: File tab.



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 →**.

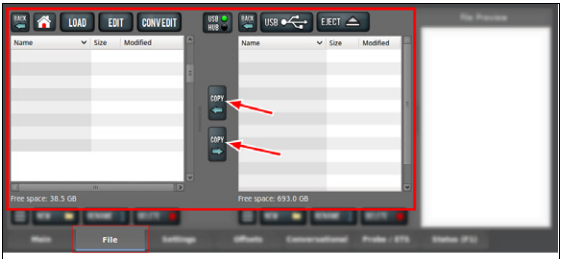


Figure 9-9: File tab.



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.

9.7 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**.

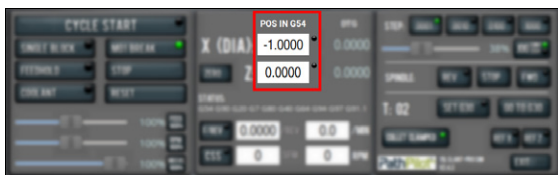


Figure 9-10: Work Offset DRO fields.

To change work offsets:

- 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.
- 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.

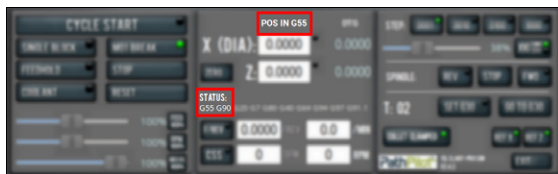


Figure 9-11: Work offset indicated in the PathPilot interface.



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).

9.7.1 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: BASIC OPERATIONS

9.8 Set Tool Geometry Offsets

9.8 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 144).

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.

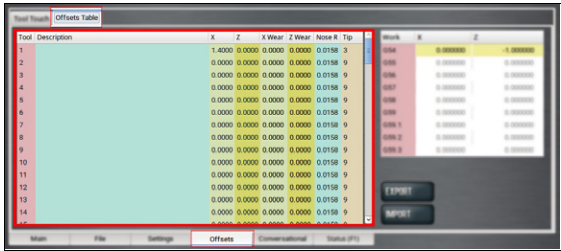


Figure 9-12: Tool Table window on the Offsets tab.

- 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.

9.9 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).
The tools along top are rear tool post tools (used by machines with a turret 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.

9.9.1 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 negative.

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

9.9.2 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**.

9.9.3 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 negative 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 negative X (diameter) values,

9: BASIC OPERATIONS

9.9 Touch Off the Tool Geometry Offsets

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

9.9.4 Import and Export the Tool Table

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

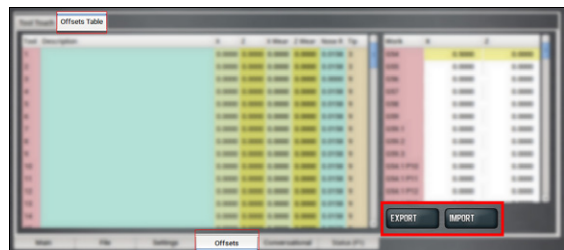


Figure 9-13: Export and Import buttons on the Offsets tab.

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.

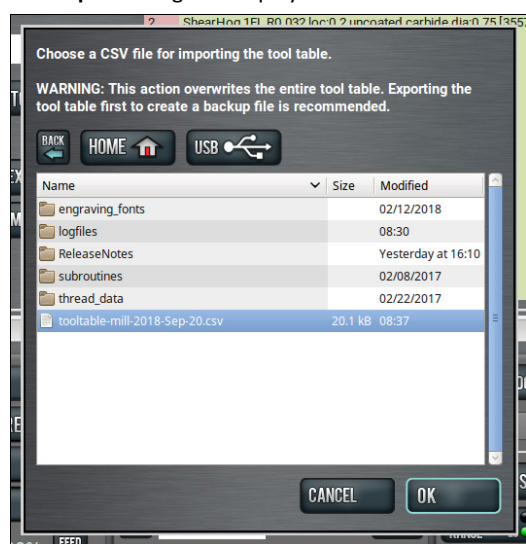


Figure 9-14: Import dialog box.

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.

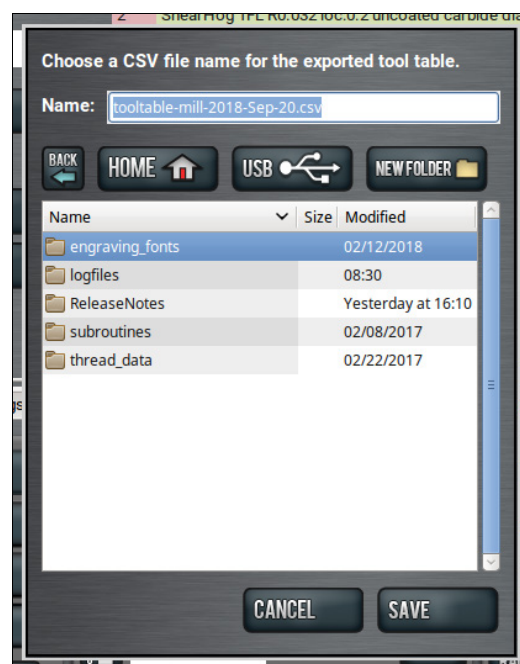


Figure 9-15: Export dialog box.

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.

9.10 OPERATE THE COOLANT PUMP

To turn coolant on or off:

- Select **Coolant**.



Figure 9-16: Coolant button.

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

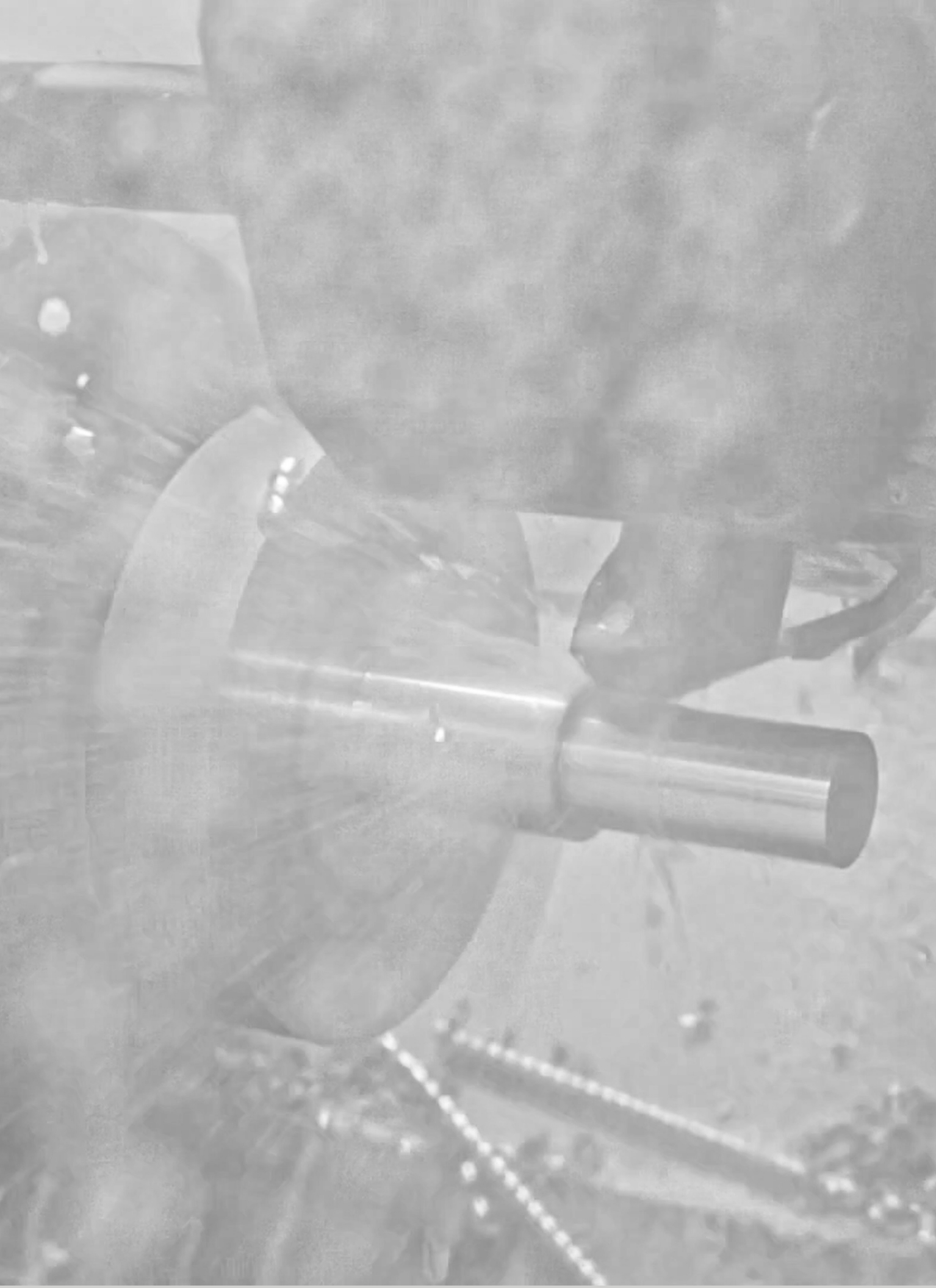
9.10.1 About Coolant

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	148
10.2 Programming Overview.....	151
10.3 Programming G-Code.....	156
10.4 Programming M-Code.....	166
10.5 Programming Input Codes.....	170
10.6 Advanced Programming.....	171

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	148
Controlled Point	148
Coordinated Linear Motion	149
Feed Rate	149
Arc Motion	149
Dwell	149
Units	149
Current Position	149
Work Offsets	149
Selected Plane	149
Tool Table	149
Wear Offsets	149
Path Control Modes	150
Feed and Speed Override Controls	150
Block Delete Control	150
Optional Program Stop Control	150

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 up and away from 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). If this tool is put in the turret, or a rear position on the gang plate, it's orientation 2.

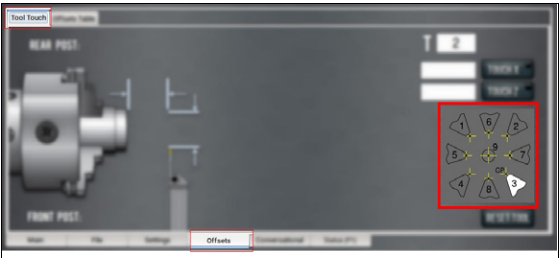


Figure 10-1: Cutting point orientation graphic.

- Tools with **orientations 1, 2, 3, and 4** 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 **orientations 6 and 8** 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 4. 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. Note that either command causes the turret to change position to pocket 2 if the machine is equipped with a turret. 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	151
10.2.2 G-Code Formatting Reference	151
10.2.3 Supported G-Codes Reference	155

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 +0.1234y 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 Note: The lathe and the RapidTurn both use G07 (X positions displayed in diameter values). G08 is not used or supported in PathPilot.
<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	156
10.3.2 Rapid Linear Motion (G00).....	156
10.3.3 Linear Motion at Feed Rate (G01).....	156
10.3.4 Arc at Feed Rate (G02 and G03).....	157
10.3.5 Dwell (G04).....	157
10.3.6 Lathe Diameter Mode (G07).....	157
10.3.7 Set Offsets (G10).....	157
10.3.8 Plane Selection (G17, G18, G19).....	159
10.3.9 Length Units (G20 and G21).....	159
10.3.10 Return to Predefined Position (G28 and G28.1) ..	159
10.3.11 Return to Predefined Position (G30 and G30.1) ..	159
10.3.12 Spindle-Synchronized Motion (G33).....	160
10.3.13 Rigid Tapping (G33.1).....	160
10.3.14 Cutter Compensation (G40, G41, G42).....	161
10.3.15 Dynamic Cutter Compensation (G41.1 and G42.1).....	161
10.3.16 Absolute Coordinates (G53).....	161
10.3.17 Select Work Offset Coordinate System (G54 to G54.1 P500).....	161
10.3.18 Set Exact Path Control Mode (G61).....	162
10.3.19 Set Blended Path Control Mode (G64).....	162
10.3.20 High Speed Peck Drill (G73).....	162
10.3.21 Threading Cycle (G76).....	162
10.3.22 Tapping Cycle (G84).....	163
10.3.23 Distance Mode (G90 and G91).....	164
10.3.24 Arc Distance Mode (G90.1 and G91.1).....	164
10.3.25 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	164
10.3.26 Feed Rate Mode (G93, G94, and G95).....	165
10.3.27 Spindle Control Mode (G96 and G97).....	165

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).....	158
Set Coordinate System (G10 L2).....	158

10: PROGRAMMING

10.3 Programming G-Code

Set Tool Table (G10 L10).....	158
Set Tool Table (G10 L11).....	158
Set Coordinate System (G10 L20).....	159

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)
```

- 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.

Troubleshooting

It's an error if:



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 141).

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 High Speed Peck Drill (G73)

The G73 cycle is intended for deep drilling with chip breaking. The retracts in this cycle break the chip but do not totally retract the drill from the hole. It's suitable for tools with long flutes which clear the broken chips from the hole.

Program: G73 X~ Z~ R~ L~ Q~

- Q~ is the delta increment along the Z-axis

The G73 cycle is as follows:

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

Troubleshooting

It's an error if:

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

10.3.21 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.

- $K\sim$ 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.

- $D\sim$ 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

- $R\sim$ 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).
- $Q\sim$ 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.
- $H\sim$ 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.
- $E\sim$ 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.
- $L\sim$ 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
- $P\sim$, $J\sim$, $K\sim$ or $H\sim$ is negative
- $E\sim$ is greater than half the drive line length

10.3.22 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 $X\sim$ $Y\sim$ $Z\sim$ $R\sim$ $P\sim$ $F\sim$

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

The G84 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.

10: PROGRAMMING

10.3 Programming G-Code

- × **Step 2:** Start the spindle forward.
- × **Step 3:** Move the Z-axis at the programmed feed rate (F~) to the Z-depth.
- × **Step 4:** Reverse the spindle.
- × **Step 5:** Dwell for the P number of seconds.
- × **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.

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
```

10.3.23 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.

10.3.24 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.

10.3.25 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3)



IMPORTANT! This is a legacy feature. Most modern programming methods don't use temporary work offsets.

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

- X~ is the X-axis coordinate
- Y~ is the Y-axis coordinate
- Z~ is the Z-axis coordinate
- A~ is the A-axis coordinate

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

The axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

Incremental distance mode (G91) has no effect on the action of G92.

When G92 is executed, it is applied to the origins of all coordinate systems (G54 through G59.3).



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**

10.3.26 Feed Rate Mode (G93, G94, and G95)

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

Inverse time is used to program simultaneous coordinated linear and coordinated rotary motion. In inverse time feed rate mode, an F word means the move should be completed in **[1/F number]** minutes.



EXAMPLE

If the F number is 2.0, the move should be completed in half a minute.

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

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

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

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

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

Troubleshooting

It's an error if:

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

10.3.27 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.....	166
10.4.2 Program Stop and Program End (M00, M01, M02, and M30).....	166
10.4.3 Display Information and Capture Images During an M00 or M01 Break.....	167
10.4.4 Spindle Control (M03, M04, and M05).....	167
10.4.5 Coolant Control (M08 and M09).....	167
10.4.6 Automatic Collet Closer Control (M10 and M11)...	167
10.4.7 Override Control (M48 and M49).....	168
10.4.8 Feed Override Control (M50).....	168
10.4.9 Spindle Speed Override Control (M51).....	168
10.4.10 Set Current Tool Number (M61).....	168
10.4.11 Set Output State (M64 and M65).....	168
10.4.12 Call Subroutine (M98).....	168
10.4.13 Return from Subroutine (M99).....	169
10.4.14 USB Camera Control (M301, M302, M303).....	169

10.4.1 Supported M-Codes Reference

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

M-Code	Description
M98	Call subroutine
M99	Return from subroutine/repeat
<u>M301</u> , <u>M302</u> , <u>M303</u>	USB camera control

10.4.2 Program Stop and Program End (M00, M01, M02, and M30)

To stop a running program temporarily, regardless of the optional stop switch setting, program: M00

To stop a running program temporarily, but only if the optional stop switch is on, program: M01

It's okay to program M00 and M01 in MDI mode, but the effect probably won't be noticeable because normal behavior in MDI mode is to stop after each line of input.

If a program is stopped by an M00, M01, selecting Cycle Start restarts the program at the following line of the G-code program.

To end a program, program: M02 or M30.

M02 leaves the next line to be executed as the M02 line. M30 rewinds the G-code file. These commands can have the following effects:

- Axis offsets are set to zero (like G92 . 2) and origin offsets are set to the default (like G54)
- Selected plane is set to XY (like G17)
- Distance mode is set to absolute (like G90)
- Feed rate mode is set to units per minute mode (like G94)
- Feed and speed overrides are set to on (like M48)
- Cutter Compensation is turned off (like G40)
- The spindle is stopped (like M05)
- The current motion mode is set to G01 (like G01)
- Coolant is turned off (like M09)

No more lines of code in the file are executed after the M02 or M30 command is executed. Selecting Cycle Start starts the program back at the beginning of the file.

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 106).

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 (M08 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



Tip! This is useful if you power on the system with a tool selected but the tool turret is set for a different tool to that indicated. You can set that tool number without doing a tool change operation.

Troubleshooting

It's an error if:

- Q~ is not 0 or greater

10.4.11 Set Output State (M64 and M65)



Note: These commands are only useful when the machine is equipped with the USB M-Code I/O Interface Kit.

There are four output relays available on the USB I/O module.

To activate output relays (contact close), program: M64

To deactivate output relays (contact open), program: M65

There are four contacts, numbered from 0 to 3. The contact is specified by the P word.



EXAMPLE

- Activating the first relay: M64 P0
- Activating the second relay: M64 P1

The outputs are deactivated using M65 with the P word specifying the relay.



EXAMPLE

- Deactivating the second relay: M65 P1
- Deactivating the fourth relay: M65 P3

There is only one P word and one relay per line. Each relay command must be done on an individual line.

The following is legal:

```
M64 P0
M64 P2
M64 P3
```

The following is not legal:

```
M64 P023
M64 P0 P2 P3
```

10.4.12 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.

10.4.13 Return from Subroutine (M99)

To return from a subroutine, program: [M99](#)

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

10.4.14 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 106).

10: PROGRAMMING

10.5 Programming Input Codes

10.5 PROGRAMMING INPUT CODES

Read the following sections for reference:

10.5.1 Feed Rate (F)	170
10.5.2 Spindle Speed (S)	170
10.5.3 Change Tool Number (T)	170

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 have an (optional) turret, numbers T01 through T08 automatically actuates the turret to select the chosen tool. Numbers greater than T08 are assumed to be gang tools, and the program continues after applying the relevant offsets. 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.

10.6 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	171
10.6.2 Expressions	173
10.6.3 Subroutines	174

10.6.1 Parameters

Read the following sections for reference:

Parameters Reference	171
Numbered Parameters Reference	172
Subroutine Parameters Reference	172
Named Parameters Reference	172

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

10: PROGRAMMING

10.6 Advanced Programming

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.

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.....	173
Functions Reference.....	173

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)

10: PROGRAMMING

10.6 Advanced Programming

- `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	174
Conditional Subroutines Reference	175
Repeating Subroutines Reference	175
Looping Subroutines Reference	175

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`.

Note: File names are lowercase letters only.
 `o<MyFile>` is converted to `o<myfile>` by the interpreter.

To execute a subroutine in a program, it must be called. To call a subroutine, program `o~ call` where `~` is the subroutine name. The subroutine name may be either a named file, a numbered file, or an expression that evaluates to a valid subroutine label.

- Expression example: `o[#101+2] call`
- Named file example: `o<myfile> call`
- Numbered file example: `o123 call`

`o~ call` takes up to 30 optional arguments, which are passed to the subroutine as `#1, #2, ..., #N`. Unused parameters from `#N+1` to `#30` have the same value as in the calling context.

Parameters `#1-#30` are local to the subroutine. On return from the subroutine, the values of parameters `#1` through `#30` (regardless of the number of arguments) are restored to the values they had before the call.

The following calls a subroutine with three arguments: `o200 call [1] [2] [3]`

Because `1 2 3` is parsed as the number `123`, the parameters must be enclosed in square brackets.

Subroutine bodies may be nested.

- Nested subroutines may only be called after they are defined.
- They may be called from other functions, and may call themselves recursively if it makes sense to do so.
- The maximum subroutine nesting level is 10.

Subroutines do not have return values, but they may change the value of parameters above `#30` and those changes are visible to the calling G-code. Subroutines may also change the value of global named parameters.

Conditional Subroutines Reference

Subroutines can be conditionally executed using the `if/endif` or the `if/else/elseif/endif` keyword constructs.

if/endif

The `if/endif` conditional will execute a block of code following the `if` keyword only when the `if` argument evaluates to true.

```
If/endif example:
o100 sub
```

```
(notice that the if-endif block uses a different
number)
o110 if [#2 GT 5]
(some code here)
o110 endif
(some more code here)
o100 endsub
```

if/elseif/else/endif

The `if/elseif/else/endif` conditional will execute the block of code following the `if` keyword when its argument evaluates to true. If the argument evaluates to false, then the code following each `elseif` is executed as long as the associated `elseif` argument evaluates to true. If no `elseif` keywords are present, or if all `elseif` arguments evaluate to false, then the code following the `else` keyword is executed.

```
If/elseif/endif example:
o102 if [#2 GT 5] (if parameter #2 is greater than
5 set F100)
F100
o102 elseif [#2 LT 2] (else if parameter #2 is
less than 2 set F200)
F200
o102 else (else if parameter #2 is 2 through 5 set
F150)
F150
o102 endif
```

Repeating Subroutines Reference

Subroutines can be repeated a finite number of times using the `repeat/endrepeat` keyword.

```
Repeat example:
(Mill 5 diagonal shapes)
G91 (Incremental mode)
o103 repeat [5]
... (insert milling code here)
G00 X1 Y1 (diagonal move to next position)
o103 endrepeat
G90 (Absolute mode)
```

Looping Subroutines Reference

Subroutines can be looped using the `do/while` or `while/endwhile` keyword constructs.

10: PROGRAMMING

10.6 Advanced Programming

do/while

The do/while loop executes a block of code once and continues to execute the code block until the while argument evaluates to true.

```
Do/while loop example:
#1 = 0 (assign parameter #1 the value of 0)
o100 do
(debug, parameter 1 = #1)
o110 if [#1 EQ 2]
#1 = 3 (assign the value of 3 to parameter #1)
(msg, #1 has been assigned the value of 3)
o100 continue (skip to start of loop)
o110 endif
(some code here)
#1 = [#1 + 1] (increment the test counter)
o100 while [#1 LT 3]
M02
```

- A label on break or continue which does not point to a matching while or do
- A label on endrepeat or endwhile no referring to a corresponding while or repeat

while/endwhile

The while/endwhile repeats a set of statements an indefinite number of times, as long as the while argument evaluates to true.

```
While/endwhile example:
(draw a sawtooth shape)
G00 X1 Y0 (move to start position)
#1 = 1 (assign parameter #1 the value of 0)
F25 (set a feed rate)
o101 while [#1 LT 10]
G01 X0
G01 Y[#1/10] X1
#1 = [#1+1] (increment the test counter)
o101 endwhile
M02 (end program)
```

The following statements cause an error message and abort the interpreter:

- A return or endsub not within a sub definition
- A label on repeat which is defined elsewhere
- A label on while which is defined elsewhere and not referring to a do
- A label on if defined elsewhere
- A undefined label on else or elseif
- A label on else, elseif or endif not pointing to a matching if

MACHINE MAINTENANCE

IN THIS SECTION, YOU'LL LEARN:

- About the required maintenance procedures that you must do so that this machine operates as designed.



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

CONTENTS

11.1 Maintenance Safety	178
11.2 Maintenance Schedules	179
11.3 Clean the Z-Axis Wipers	180
11.4 Clean the Coolant System	181
11.5 Examine the Enclosure Windows	182
11.6 Examine the Spindle Belt	183
11.7 Lubricate the Machine	184
11.8 Prevent Rust	185
11.9 Adjust the Gibs	186

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 207). 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.
- ☐ 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.
- ☐ Verify that the door switches are properly functioning.

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.

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.

11.2.4 Semi-Annually

- ☐ 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

- ☐ 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 Z-Axis Wipers

11.3 CLEAN THE Z-AXIS WIPERS

To extend the service life of the machine, you must make sure that the way Z-axis wipers are protecting the machine's slideways. Regularly do the following:

- Examine the way Z-axis wipers for signs of wear or damage. If the way Z-axis wipers are damaged, you must replace them.
- Use a chip brush to clean the way Z-axis wipers of chips and other swarf.

11.4 CLEAN THE COOLANT SYSTEM

To extend the service life of the coolant and the coolant pump, you must make sure that both are free of contaminants.

Regularly do the following:

- Use a chip brush to clean the machine, chip trays, and enclosure of chips and other swarf.
- Empty the chip strainer on the coolant tank.
- Measure the coolant concentration, pH, and volume, and then compare your findings to the material's safety data sheet (SDS).
 - **Coolant Level Too Low** Fill the coolant tank with fresh coolant, making sure that it is the appropriate concentration.
 - **Coolant pH Outside of Manufacturer's Recommended Coolant Limits** Adjust the pH according to the manufacturer's recommendation.



Note: You must only use coolant that is recommended for machining. See "Cutting Fluid Reference" (below).

- Examine the coolant pump's impeller for blockages.
- To remove tramp way oil from the coolant tank, use one of the following:
 - Oil Skimmer Kit (PN 39298)
 - Floating Tramp Oil Collection Pillow (PN 31925)

Periodically do the following:

- Remove the coolant tank covers and the chip strainer, and then clean the fine swarf from the coolant tank.
- If you're using a filter to prevent small particles from entering the coolant tank, replace it when it's saturated with chips, swarf, or tramp oil.
- Examine the coolant for contamination, and determine if the coolant's oil-water emulsion has broken.
- Drain and thoroughly clean the coolant tank, and then replace with fresh coolant.

NOTICE! Hazardous Waste: Only dispose of coolant as advised by local environmental protection authorities. You must not dispose of coolant in a septic system or a sewer.

11.4.1 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.

11.4.2 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

11.5 Examine the Enclosure Windows

11.5 EXAMINE THE ENCLOSURE WINDOWS

Regularly examine the enclosure windows, and all other guarding, for signs of wear or damage.

When required, replace the windows. Refer to "Enclosure Assembly" (page 232) to view the enclosure's diagrams and parts lists.

11.6 EXAMINE THE SPINDLE BELT

To extend the service life of the spindle belt, you must regularly examine it for signs of damage or wear.

If the spindle belt is damaged, you must replace it with Timing Belt, HTD-808-8M-25 (PN 54318).

11: MACHINE MAINTENANCE

11.7 Lubricate the Machine

11.7 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).

11.7.1 About the Automatic Oiler

The automatic oiler is powered whenever the machine is powered on, and has the following default settings:

- Lubrication interval: 480 minutes (8 hours)
- Actuation time: 12 seconds

11.7.2 Lubrication System Reference

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

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

11.8 PREVENT RUST

Take proper care to protect all exposed iron and steel surfaces on your machine. To reduce the possibility of rust, you must regularly do the following:

- Clean all exterior surfaces with a mild cleaner.
- Only use flood coolant that is recommended for machining (see "Cutting Fluid Reference" (page 181)).
- 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).

11.8.1 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.

11: MACHINE MAINTENANCE

11.9 Adjust the Gibs

11.9 ADJUST THE GIBS

This procedure describes how to adjust the gibs on both axes. The Z-axis is described in detail. The gib plate is tapered from right to left. When the gib plate is moved in the Z negative direction (to the left of the operator), the Z-axis saddle the slide way tightens; when it moves in the Z positive direction, the slideway loosens. With the machine is powered on, move the Z-axis to near the middle of its travel and locate the gib plate screw on either end of the Z saddle.

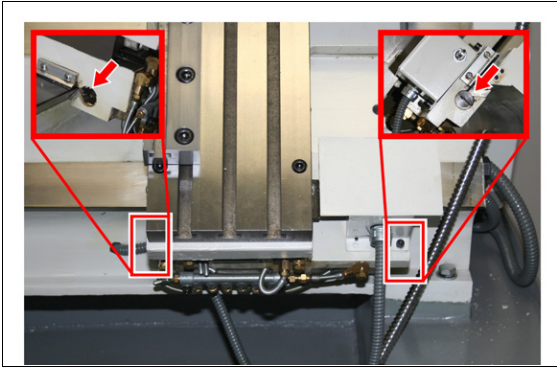


Figure 11-1: Gib plate screws on either side of the Z saddle.

It is difficult to assess the correct clearance for the gib as a very small force on its end can, via the shallow taper, apply a large force to the dovetail. Too tight a gib results in zero gap and very high friction since there is no longer space for an oil film.

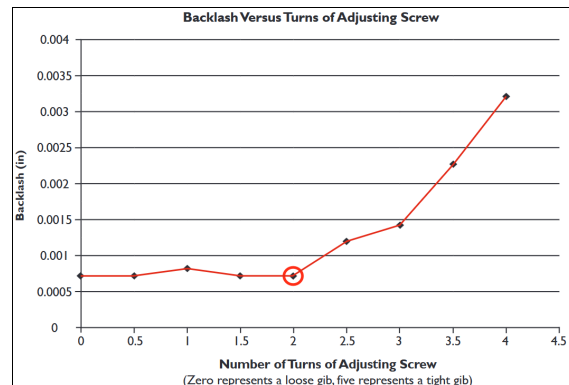
The easiest method is by a form of dead reckoning:

1. Ensure the left (loosening) gib screw is unscrewed by several turns to allow for clearance for the gib to be adjusted to the left.
2. Tighten the right (tightening) gib screw in, thus pushing the gib left until it is tight yet not hitting the bottom screw. Wiggle the carriage while you are tightening this screw to get the gib reasonably tight – but not so tight as to strip the screw head or flex the castings.
3. Loosen the right screw four full turns.
4. Tighten the left screw, thus pushing the gib back up against the upper screw and creating a positive amount of gib clearance. The gib should be held in place with upper and lower screw tight on the gib.

IMPORTANT! Ensure both adjustment screws are tight after any gib adjustment. Failure to do so will result in gib movement.

A more precise method of adjusting the gibs is to use lost motion, apparent as backlash on a dial indicator, to arrive at the correct setting:

1. Loosen the right screw eight turns and tighten the left screw eight turns so that the gib clearance is loose.
2. Use a dial indicator to measure the axis backlash. With a very loose gib, the measured backlash is entirely attributable to the backlash in the belt reduction drive, the angular contact bearings, and the ball nut. On a newer machine, the value should be under 0.0025" on either axis.
3. Tighten the gib by one turn (by loosening the lower screw first, then tightening the upper screw). Measure the backlash again.
4. Repeat this procedure until the measured backlash increases – this is the point at which the gib setting has started to induce lost motion in the axis.
5. Back the adjustment off to the point just before you saw the increased backlash. That is the ideal setting for the axis.



Note: For the X-axis, the gib tightens with the gib screw on the bottom (operator side). No covers need to be removed.

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.....	188
12.2 Getting Help.....	189
12.3 Required Tools.....	190
12.4 Frequently Found Problems.....	191
12.5 Electrical Service.....	192
12.6 Power Distribution Subsystem.....	194
12.7 Control Power Subsystem.....	196
12.8 Servo Drive Subsystem.....	197
12.9 Spindle Drive Subsystem.....	202
12.10 Servo IO Breakout Board Subsystem.....	205



TORMACH®

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)

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.

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:

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. Remove the power plug(s) from the wall outlet. If your system is hardwired, isolate the machine by opening its circuit breaker(s).
 - e. 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. Remove the power plug(s) from the wall outlet. If your system is hardwired, isolate the machine by opening its circuit breaker(s).
 - e. Follow correct lockout/tagout procedures.
2. Unplug and firmly reseal connectors.

12: TROUBLESHOOTING

12.5 Electrical Service

12.5 ELECTRICAL SERVICE

12.5.1 LED Identification

 **WARNING!** Electrocution 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.

Many electrical problems are self-apparent, and you can trace them by observing LEDs in PathPilot, LEDs in the electrical cabinet, and the machine's actions.

LEDs in PathPilot indicate output or functional status — including the LEDs on the Status tab, which are useful for indicating if any inputs or outputs are operational.

There are various LED indicators in the electrical cabinet. Among these are:

- IO Control Board Indicators**

Category	Function	LED Identifier
General	Control Power	D37
	Output Power	D38
	E-Stop Status	D39
Input Status	Accessory Input 1	D20
	Accessory Input 2	D21
	Accessory Input 3	D22
	Accessory Input 4	D23
	Enclosure Lock Switch State	D28
	Enclosure Button State	D29
	Collet Closer Foot Pedal	D30
	Spindle Door Switch	D34
Output Status	Flood Coolant	D40
	Washdown Coolant	D41
	X-Axis Brake	D52
	Enclosure Door Lock LED	D44
	Accessory Output 1	D45
	Enclosure Door Unlock	D54
	Accessory Output 2 (Collet Closer)	D50
	Spindle Drive Contactor Start Pulse	D51

- Axis Indicators on the X-, Z-, and A-Axis Drivers** Each Driver has a five-digit LED display used to show the servo's status, parameter settings, and error codes.

- **PS1 24 Vdc Power Supply** The power supply has a power on LED.
- **S1 Door Latch Push Button** The push button has an integrated status light that shows the machine is in an operating state where unlatching the door is allowed.
- **PathPilot Operator Console** Console has an integrated RGB status light that indicates the current machine operating state or fault condition.

12.5.2 Overview

The 15LX electrical system consists of several interrelated subsystems:

- **230 Vac Power Distribution** Provides 230 Vac mains power to servo drives. Also provides pass-through power for accessory devices (such as flood and washdown coolant pumps, the PathPilot console, turret, and automatic oiler) and auxiliary devices (such as enclosure lights).
- **24 Vdc Safety and Control** Provides 24 Vdc power to the Servo IO breakout board, electrical cabinet and fans, as well as output devices powered through the Servo IO breakout board via the board's J2 connector.
- **Servo Drives** The three servo drives power and control the machine's spindle motor and two axis motors. Each servo drive includes motor power and encoder connections, a dedicated IO connection, and a Safe Torque Off (STO) connection. The STO functionality allows the drive to be safely disabled in the event of an Emergency Stop or machine fault. Additionally, each servo drive communicates with the PathPilot controller using an EtherCAT® fieldbus connection. Accessories such as the turret include additional servo drives that are connected to the EtherCAT fieldbus system. The servo drives are also capable of reporting diagnostic information and error codes through the PathPilot controller via the EtherCAT fieldbus system.
- **Servo IO Breakout Board** The machine's main control board is the primary interface between the individual servo drives' input and output lines and the sensors and devices connected to them. The Servo Breakout Board connects to the X, Z, turret and Spindle servo drives via the IO-7, IO-10, IO-4, and IO-12 cables respectively. The Servo Breakout Board has additional connectors for the Emergency Stop/Reset push buttons, power drawbar system, and enclosure door latch system as well as connectors for various accessories (including the automatic collet closer and collet closer foot pedal system).

12: TROUBLESHOOTING

12.6 Power Distribution Subsystem

12.6 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, major accessories, including the flood and washdown coolant pumps, and the PathPilot controller.

To troubleshoot the power distribution subsystem, read the following:

12.6.1 The Machine Won't Power On	194
12.6.2 The PathPilot Operator Console Won't Power On	194

12.6.1 The Machine Won't Power On

Cause: The machine isn't plugged in to an outlet.		
Probability	How-To Steps	Need More?
High	➤ Reseat the NEMA L6-30 plug into an outlet.	If needed, measure for 208 Vac-240 Vac between wires 1001 and 1002/N at the DISC1 Main Disconnect.
Cause: The facility's mains breaker tripped.		
Probability	How-To Steps	Need More?
High	➤ Examine power cable for damage or exposed wires, and then reset your facility's circuit breaker serving the machine.	If needed measure for 208 Vac-240 Vac between wires 1001 and 1002/N at the DISC1 Main Disconnect.
Cause: The CB1 circuit breaker tripped.		
Probability	How-To Steps	Need More?
High	➤ If the machine, controller, accessories, and enclosure lights don't have power, examine the power cord(s) and wires for damaged or exposed wires, and then reset the CB1 circuit breaker.	If needed measure for 208 Vac-240 Vac between wires 1005 and 1006/N at the CB1 circuit breaker.
Cause: The Main Disconnect switch is in the OFF position.		
Probability	How-To Steps	Need More?
High	➤ Switch DISC1 Main Disconnect to the ON position.	If needed, measure for 208 Vac-240 Vac between wires 1003 and 1004/N at the DISC1 Main Disconnect.

12.6.2 The PathPilot Operator Console Won't Power On

Cause: The PathPilot operator console isn't plugged in to an outlet.		
Probability	How-To Steps	Need More?
High	➤ Reseat the power cord connection at both the electrical cabinet outlet (XS26) and the controller.	It's possible that the power cord could become loose from movement. If needed, measure for 208 Vac-240 Vac between wires 1038 and 1046/N at the XS26 Controller/Monitor outlet.

Cause: The CB4 circuit breaker tripped.

Probability	How-To Steps	Need More?
Medium	➤ If the operator console, automatic oiler, flood coolant pump, and washdown coolant pump don't have power, examine the power cords and wires for damaged or exposed wires, and then reset the CB4 circuit breaker.	CB4 affects the operator console, automatic oiler, flood coolant pump and washdown coolant pump. If needed, measure for 208 Vac-240 Vac between wires 1013 and 1014/N at the CB4 circuit breaker.

12: TROUBLESHOOTING

12.7 Control Power Subsystem

12.7 CONTROL POWER SUBSYSTEM

To troubleshoot the control power subsystem, read the following:

12.7.1 The Machine Won't Power On 196

12.7.1 The Machine Won't Power On

Cause: The Emergency Stop button is pushed in.

Probability	How-To Steps
High	➤ Twist out the Emergency Stop button and press the Reset button.

Cause: The DC power supply PS1 is defective.

Probability	How-To Steps	Need More?
Medium	• Check if Electrical Cabinet Fan (FAN1) is running. • Check for DC OK LED on PS1 DC power supply. • Check for Control Power (D37) LED on E1 Servo IO Breakout Board. • Measure for 208 Vac-240 Vac between wires 1025 and 1026/N at the PS1 DC power supply. • Measure for 24 Vdc nominal between wires 4001 and 4002 at the PS1 DC power supply.	Before you replace the DC power supply, power off the machine (see "Power off the Machine" (page 55)).

Cause: The DC power supply PS1 output voltage is too low.

Probability	How-To Steps
Low	• Check if Electrical Cabinet Fan (FAN1) and is running. • Check for DC OK LED on PS1 DC power supply. • Measure for 208 Vac-240 Vac between wires 1025 and 1026/N at the PS1 DC power supply. • Measure for 24 Vdc nominal between wires 4001 and 4002 at the PS1 DC power supply. • If output voltage is less than 24 Vdc, adjust the output voltage using the +V ADJ trim potentiometer on the front of the PS1 DC power supply.

Cause: The CB1, CB2, or CB4 circuit breaker tripped.

Probability	How-To Steps	Need More?
Low	➤ Examine the power cords and wires for damaged or exposed wires, and then reset the circuit breaker.	If needed, measure for 208 Vac-240 Vac between wires 1005 and 1006/N at the CB1 circuit breaker, between wires 1021 and 1022/N at the CB2 circuit breaker, between wires 1019 and 1020/N at the CB3 circuit breaker, and between wires 1017 and 1018/N at the CB4 circuit breaker.

12.8 SERVO DRIVE SUBSYSTEM

The servo drives provide power and control to the X- and Z-axis motors as well as the spindle motor and the turret motor. The servo drives communicate with PathPilot via the EtherCAT fieldbus connection. Each servo drive also handles a dedicated group of IO connections.

To troubleshoot the axes drive subsystem, read the following:

12.8.1 All Axes Won't Move When Commanded	197
12.8.2 One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move	198
12.8.3 Axis Movement is Noisy	199
12.8.4 Lost Motion on Axis Travel	200

12.8.1 All Axes Won't Move When Commanded

Cause: Control signals aren't reaching the servo motors.

Probability	How-To Steps	Need More?
High	1. Power off the machine (see "Power off the Machine" (page 55)). 2. Examine the EtherCAT® connectors at each servo drive. All servo drives must be connected for the machine to function. Verify that the connectors are firmly seated, and examine the connectors for loose or damaged pins.	Examine the DATA-1, DATA-2, DATA- 5, and optional DATA-8 cables connecting the servo drives. Also examine the Ethernet cable connecting the ETH1 Machine Control Connector to the PathPilot controller.

Cause: PathPilot isn't commanding the move due to a Hold-to-Run condition.

Probability	How-To Steps
High	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, using hold to run is required while the enclosure door is open. 2. Close the enclosure door. 3. From the PathPilot interface, select the Main tab, then press the Esc key on the keyboard. 4. On the keyboard, press and hold one of the Arrow keys to jog an axis and examine the value displayed in its DRO field. 5. Using the jog pendant, select either the X- or Z-axis, then turn the jog wheel, and examine the value displayed in its DRO field. 6. Open the enclosure door. On the keyboard, press and hold an Arrow key to jog an axis. Using the arrow keys jogs the axis and functions as a hold-to-run button. 7. With the enclosure door open, press and hold the Hold-to-Run button on the side of the jog pendant. Then turn the jog wheel and examine the value displayed in the DRO field. 8. If the problem persists, insert the key into the top of the operator console and switch to Setup mode. Then, repeat Steps 1 through 7. 9. If the problem persists, there may be a problem with the controller. Restart the controller. Then, repeat Steps 1 through 8.

12: TROUBLESHOOTING

12.8 Servo Drive Subsystem

Cause: PathPilot isn't commanding the move, or there's a controller problem.

Probability	How-To Steps
Low	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.

12.8.2 One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move

Cause: The servo drive has an error present.

Probability	How-To Steps
Low	1. Examine the servo drives in the electrical cabinet for an error code. 2. Power the machine on and off, and the trip should reset. 3. If the problem continues, 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.

Cause: There's a loose axis motor coupling.

Probability	How-To Steps
Low	• Jog the axis and listen to determine if you can hear the motor run. • Examine the coupling and make witness marks to determine if the motor's turning but the screw isn't.

Cause: Oil isn't getting to the ways and ball screw.

Probability	How-To Steps
Low	1. Examine the oil level in the oiler. 2. Investigate the oiling system for lack of oil and/or plugged lines.

Cause: There's oil residue from long-term storage.

Probability	How-To Steps
Low	➤ Repeatedly pump oil and slowly jog the axis.

Cause: There's debris on the ball screw.

Probability	How-To Steps	Need More?
Low	➤ 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).

12.8.3 Axis Movement is Noisy

Before You Begin

⚠ WARNING! Electrocution 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 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.

Cause: There's a loose wire connection or failed connector on a driver.

Probability	How-To Steps	Need More?
High / Low	➤ Power off the machine (see "Power off the Machine" (page 55)). Then, check all driver connections.	With the machine powered off, examine the power connections to L1, L2, L1C and L2C on the servo drives for signs of overheating.

Cause: There's a defective servo motor.

Probability	How-To Steps	Need More?
Medium	➤ See "One Axis Won't Move (or Only Moves in One Direction), and Other Axes Move" (on the previous page).	There have been cases of a noisy axis relating to a defective servo motor. This may be temperature-dependent.

12: TROUBLESHOOTING

12.8 Servo Drive Subsystem

12.8.4 Lost Motion on Axis Travel

Cause: Improper use of tool offset, work offset, or cutter compensation.

Probability	How-To Steps	Need More?
High	➤ 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 is operator error.

Cause: The motor coupling is loose or cracked.

Probability	How-To Steps	Need More?
Low	➤ 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: The X-axis brake isn't releasing.

Probability	How-To Steps	Need More?
Low	➤ Slowly jog the X-axis: if the brake isn't releasing, the X-axis will usually move down properly, but won't move up. A faulty X-axis brake can over-torque and fault the X-axis motor.	You should be able to hear the motor whenever you command the axis to move. Usually, the brake alone does not have the torque to cause a loss of motion. A condition like poor lubrication in combination with a defective X brake are required to actually lose position.

Cause: Controller or PathPilot problem.

Probability	How-To Steps	Need More?
Low	➤ 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 189).

Cause: There's an obstruction or excessive friction, or there's high load in the mechanical system.

Probability	How-To Steps	Need More?
Low	➤ Jog the axis and carefully observe the motion.	Typical mechanical issues include: an increase in friction due to lack of oil at the linear way surfaces or ball screw. They also come from excessive load on the system due to chips or debris a sticking X-axis brake or an end-of-travel bumper wedged against the motor mount casting.

12: TROUBLESHOOTING

12.9 Spindle Drive Subsystem

12.9 SPINDLE DRIVE SUBSYSTEM

The machine's spindle is driven by .
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:

12.9.1 The Spindle Won't Turn	202
12.9.2 Machining Operations are Loud ("Chatter")	204

12.9.1 The Spindle Won't Turn

Cause: The motor runs, but the spindle doesn't turn.		
Probability	How-To Steps	Need More?
Low	➤ Examine the spindle belt for wear or damage.	If the display on the servo drive is on, the belt may be loose or broken. You'll hear the spindle motor running.
Cause: There's no power to the spindle servo drive.		
Probability	How-To Steps	Need More?
—	➤ Examine the servo drive display: it has power if its indicator lights are on.	When power is removed, the servo drive display remains active until the internal capacitors dissipate their energy. That usually takes about five seconds.

Cause: There's no power to the servo drive output stage.

Probability	How-To Steps
High	There are loose power or control wires in the K1 contactor circuit. 1. Power off the machine (see "Power off the Machine" (page 55)). 2. Examine the circuit for loose wires. 3. Power on the machine (see "Power on the Machine" (page 52)) and reset the machine. Then, test operation. 4. Check for 24 Vdc between wires 4008 and 4053 at the K1 contactor coil.
Probability	How-To Steps
Low	Thermal switch (TS1) tripped, preventing K1 from latching. 1. Power off the machine (see "Power off the Machine" (page 55)). 2. Allow the brake resistor to cool, and reset thermal switch by pressing reset button (between its two terminals). 3. Power on the machine (see "Power on the Machine" (page 52)) and reset the machine. Then, test operation. 4. Check for 24Vdc between wires 4008 and 4053 at the K1 contactor coil.
Probability	How-To Steps
Low	The control board isn't providing a run command or holding the K1 contactor on. 1. Check D57 Spindle Drive Contactor Start Pulse LED on the Servo IO Breakout Board as you command a spindle rotation through PathPilot. 2. Check for 24 Vdc between wires 4051 and 4053 at the J33 connector on the Servo IO Breakout Board.

Cause: The servo drive tripped.

Probability	How-To Steps
Low	➤ If the servo drive tripped, check the error code and troubleshooting steps on the PathPilot Status screen.

Cause: The belt is loose or broken, or sheaves are not fixed to the motor or the spindle.

Probability	How-To Steps
Low	➤ Power off the machine (see "Power off the Machine" (page 55)). Then, examine the mechanical system.

Cause: The motor is defective.

Probability	How-To Steps	Need More?
Low	1. Power off the machine (see "Power off the Machine" (page 55)). 2. Wait 30 seconds and check for voltage, and then remove wires 2001, 2002, and 2003 from the servo drive's power connector. 3. Measure the resistance between: • Wires 2001 and 2002 • Wires 2002 and 2003 • Wires 2003 and 2001 Resistance should be in the range of approximately 2-4 Ω. 4. Reconnect wires to the appropriate servo drive power connector terminals.	• 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 servo drive.

12: TROUBLESHOOTING

12.9 Spindle Drive Subsystem

12.9.2 Machining Operations are Loud ("Chatter")

Cause: Inappropriate cutting parameters or CAM settings.

Probability	How-To Steps
High	1. Verify that you're using the correct cutting parameters: use the feed rate and spindle overrides to adjust cutting parameters during operation. 2. Confirm that tool offsets are correct.

Cause: Worn or broken cutting tools.

Probability	How-To Steps
High	➤ Inspect for worn or broken cutting tools, and replace as necessary.

Cause: Swarf in the spindle or on the 5C collet body.

Probability	How-To Steps
Medium	1. Inspect inside the spindle and on the 5C collet for swarf. If necessary, clean the components. 2. Verify that the spindle and 5C collet contact surfaces are clean, dry, and free of grease or oil.

12.10 SERVO IO BREAKOUT BOARD SUBSYSTEM

The machine's E1 Servo IO Breakout Board is the primary interface between the individual servo drives' input and output lines and the sensors and devices connected to them.

To troubleshoot the spindle drive subsystem, read the following:

12.10.1 Groups of Inputs and Outputs Aren't Functioning 205

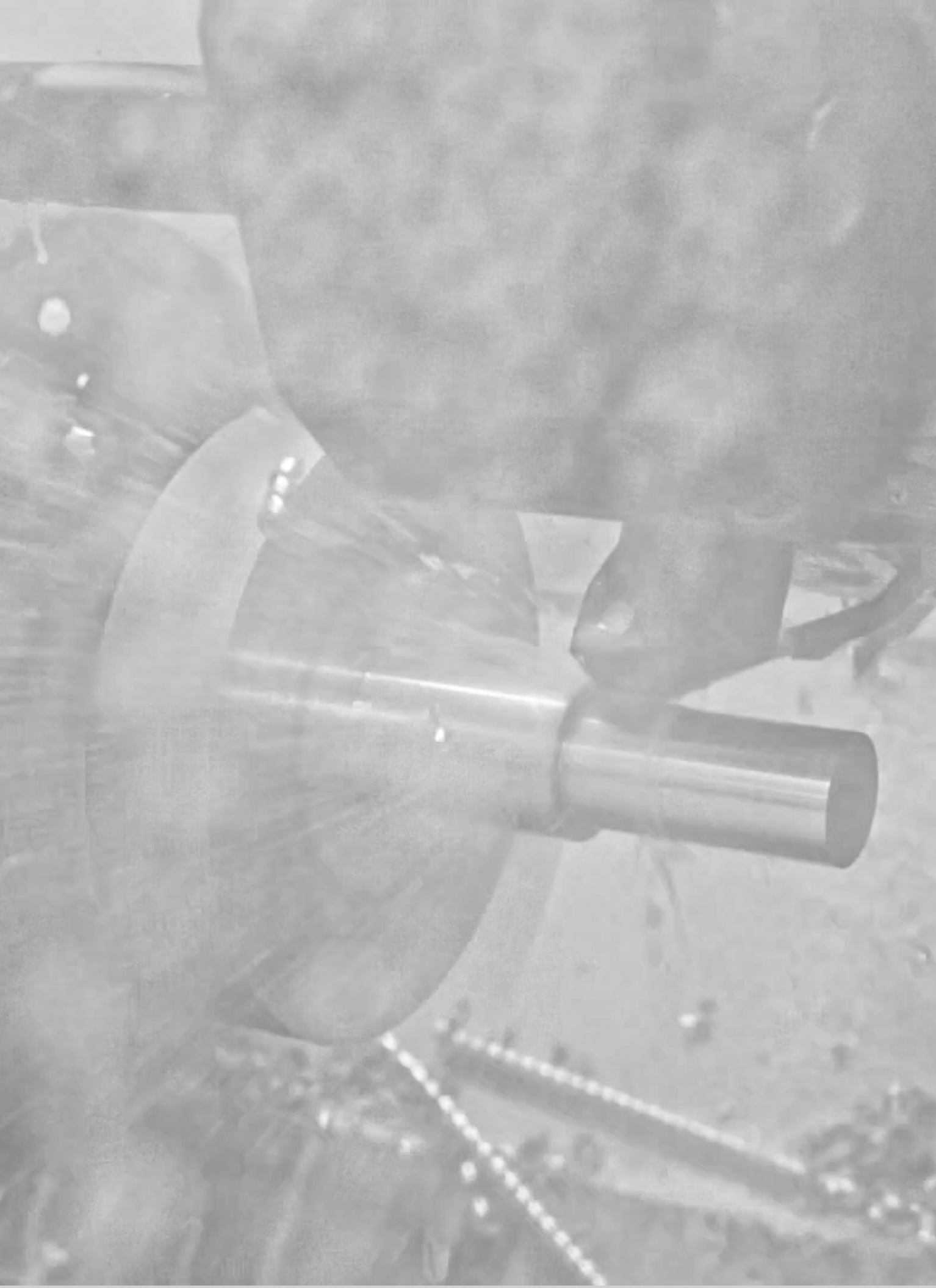
12.10.1 Groups of Inputs and Outputs Aren't Functioning

Cause: Loose IO cable between servo drive and servo IO breakout board.

Probability	How-To Steps	Need More?
Medium	- Each servo drive is responsible for a group of dedicated inputs and outputs. - Use the "Servo Drive Inputs and Outputs" (below) table to identify which drive is responsible for the non-functional inputs and outputs. - Check the identified IO cable and its connections at the Servo IO Breakout Board and Servo Drive.	

Servo Drive Inputs and Outputs

Identifier	Servo Drive	IO Cable Identifier	Inputs	Outputs
DR1	Spindle	IO-4	Spindle Door Switch	Collet Closer Spindle Contactor Start Pulse
DR2	X-Axis	IO-7	Accessory Input 1 Accessory Input 2 Accessory Input 3 Accessory Input 4	Flood Coolant Washdown Coolant X-Axis Brake
DR3	Z-Axis	IO-10	Enclosure Lock Switch State Enclosure Button State Collet Closer Foot Pedal	Enclosure Door Lock LED Accessory Output 1 Door Unlock
DR4	Turret	IO-12		



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.

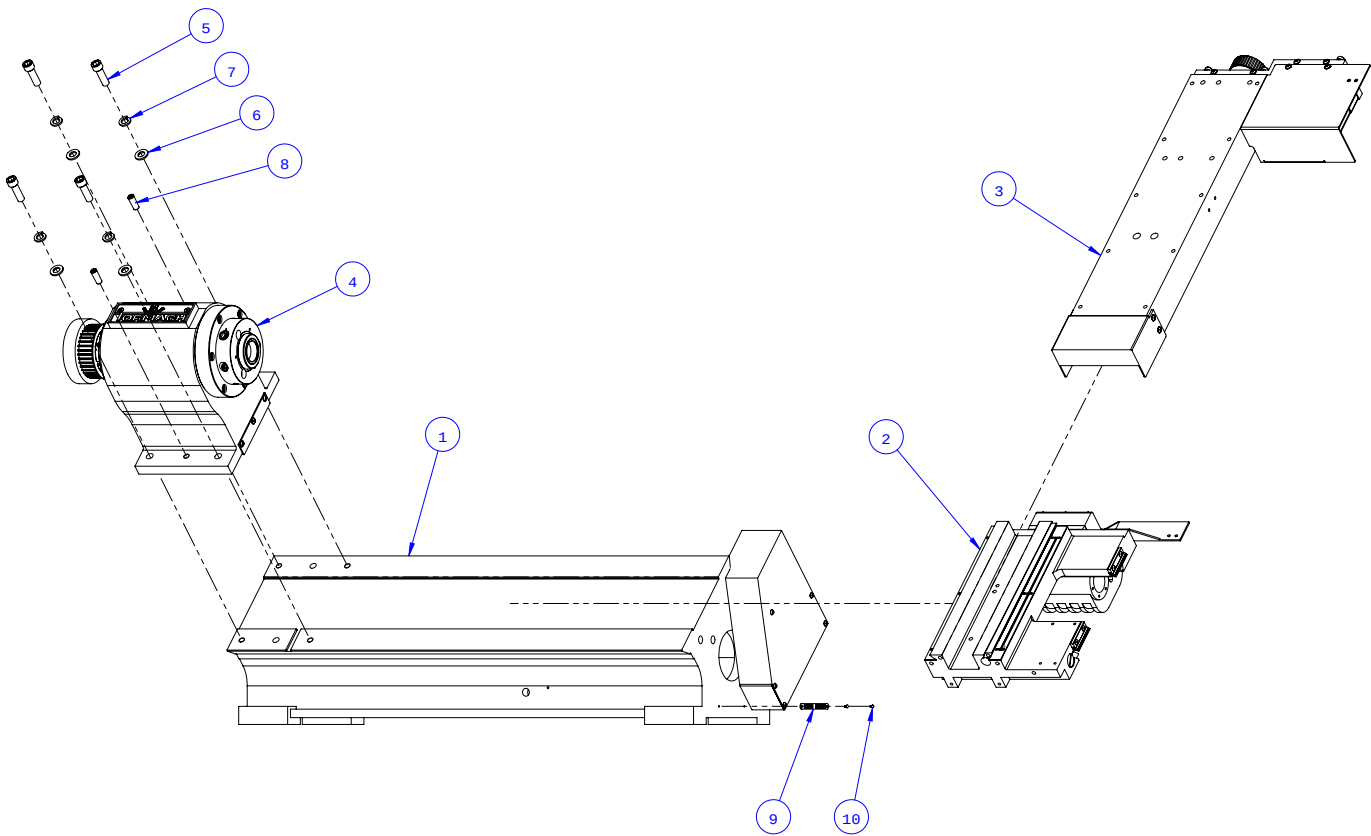
CONTENTS

13.1 Base Machine Assembly	208
13.2 Machine Base (with Motors).....	210
13.3 Bed Casting Assembly.....	212
13.4 Z-Axis Saddle Assembly.....	214
13.5 X-Axis Cross Slide Assembly.....	216
13.6 Headstock Assembly.....	218
13.7 Stand Assembly.....	221
13.8 Electrical Cabinet Assembly.....	225
13.9 Machine Pneumatics Distribution Assembly.....	230
13.10 Enclosure Assembly.....	232
13.11 Flood Coolant Kit Assembly.....	242
13.12 Accessory Diagrams and Parts Lists.....	248

13: DIAGRAMS AND PARTS LISTS

13.1 Base Machine Assembly

13.1 BASE MACHINE ASSEMBLY



13: DIAGRAMS AND PARTS LISTS

13.1 Base Machine Assembly

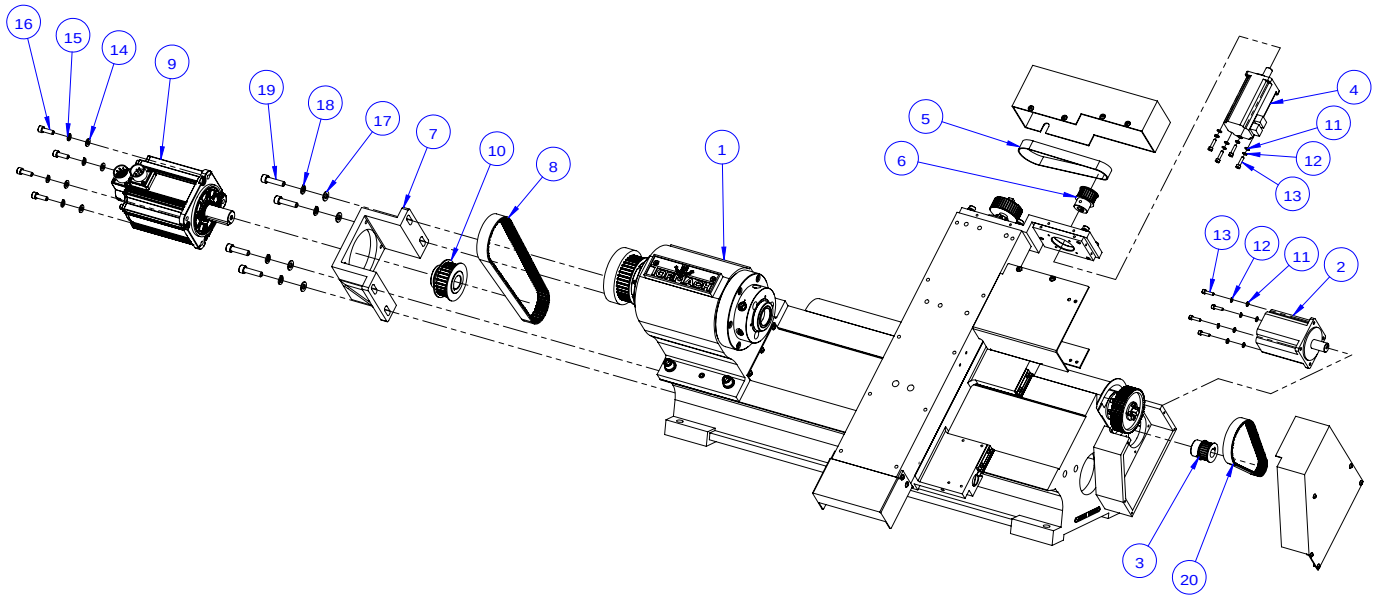
ID	Description	Alternative Part Number	Quantity
1	15LX Lathe Bed Casting Assembly (PN 54371)	—	1
2	15LX Z-Axis Saddle Assembly (PN 54372)	—	1
3	15LX Lathe Cross Slide Assembly (PN 54373)	—	1
4	Headstock Assembly, 15LX (PN 54374)	—	1
5	Screw, Socket Head Cap, M12 × 1.75 - 40 (PN 34268)	McMaster Carr: 91290A622	4
6	Washer, Flat, M12 (PN 32473)	McMaster Carr: 91166A290	4
7	Washer, Split Lock, M12 (PN 30321)	—	4
8	Taper Pin, 10 mm × 30 mm, M6 Tapped (PN 34271)	—	2
9	Subassembly Part Number/Serial Number Tag, 15LX (PN 54693)	—	1
10	Screw Nail, #4 × 0.188 in., Stainless Steel (PN 51717)	McMaster Carr: 91654A105	2

13: DIAGRAMS AND PARTS LISTS

13.2 Machine Base (with Motors)

13.2 MACHINE BASE (WITH MOTORS)

Note:
Parts without balloons are included in the exploded views of 53881 and its sub assemblies.



13: DIAGRAMS AND PARTS LISTS

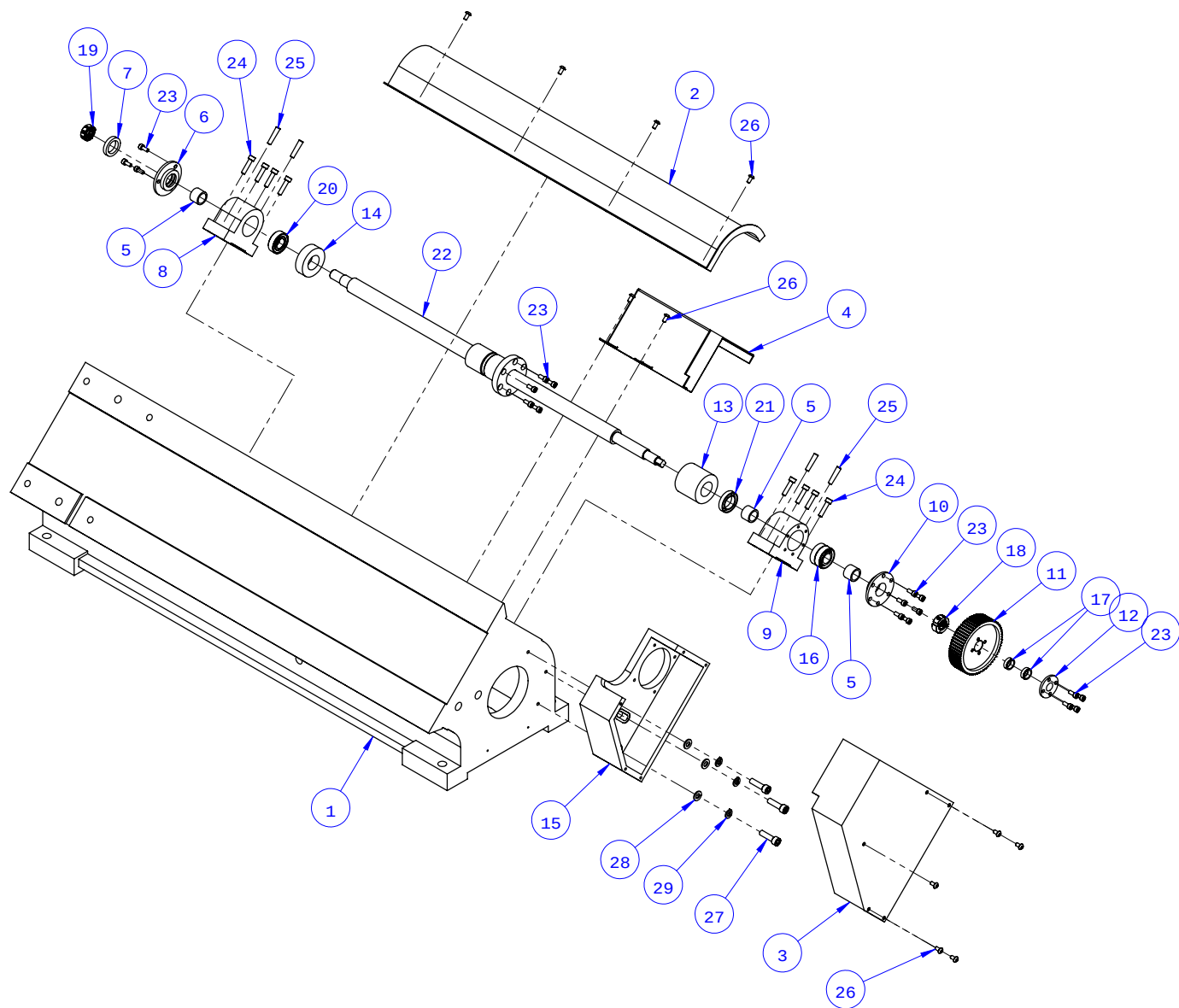
13.2 Machine Base (with Motors)

ID	Description	Alternative Part Number	Quantity
1	15LX Lathe, Base Machine Assembly (PN 53881)	–	1
2	Inovance MS1 Servo Motor (PN 54299)	–	1
3	5M-HTD-24T Pulley, 19 mm Bore, 21 mm Height (PN 54295)	–	1
4	Inovance MS1 Servo Motor (PN 54297)	–	1
5	Timing Belt, HTD-475-5M-20 (PN 33228)	–	1
6	5M-HTD-24T Pulley, 14 mm Bore, 21 mm Height, 5 mm Keyway (PN 34238)	–	1
7	15LX Spindle Motor Mounting Casting (PN 54296)	–	1
8	Timing Belt, HTD-808-8M-25 (PN 54318)	–	1
9	MS1H2-30C30CB-A331R (3 kW motor with 130 × 130 frame) (PN 52544)	–	1
10	8M-HTD-28T Pulley, 28 mm Bore, 27 mm Height, 8 mm Keyway (PN 54317)	–	1
11	Washer, Flat, M5 (PN 39305)	McMaster Carr: 91166A240	8
12	Washer, Split Lock, M5 (PN 31572)	–	8
13	Screw, Socket Head Cap, M5 × 0.8 - 20 (PN 30357)	McMaster Carr: 91290A242	8
14	Washer, Flat, M8 (PN 30531)	McMaster-Carr: 91166A270	4
15	Washer, Split Lock, M8 (PN 32339)	–	4
16	Screw, Socket Head Cap, M8 × 1.25 - 25 (PN 34635)	McMaster-Carr: 91290A432	4
17	Washer, Flat, M10 (PN 30919)	McMaster Carr: 91166A280	4
18	Washer, Split Lock, M10 (PN 31684)	–	4
19	Screw, Socket Head Cap, M10 × 1.5 - 40 (PN 30366)	McMaster Carr: 91290A528	4
20	Timing Belt, HTD-450-5M-20 (PN 54962)	–	1

13: DIAGRAMS AND PARTS LISTS

13.3 Bed Casting Assembly

13.3 BED CASTING ASSEMBLY



13: DIAGRAMS AND PARTS LISTS

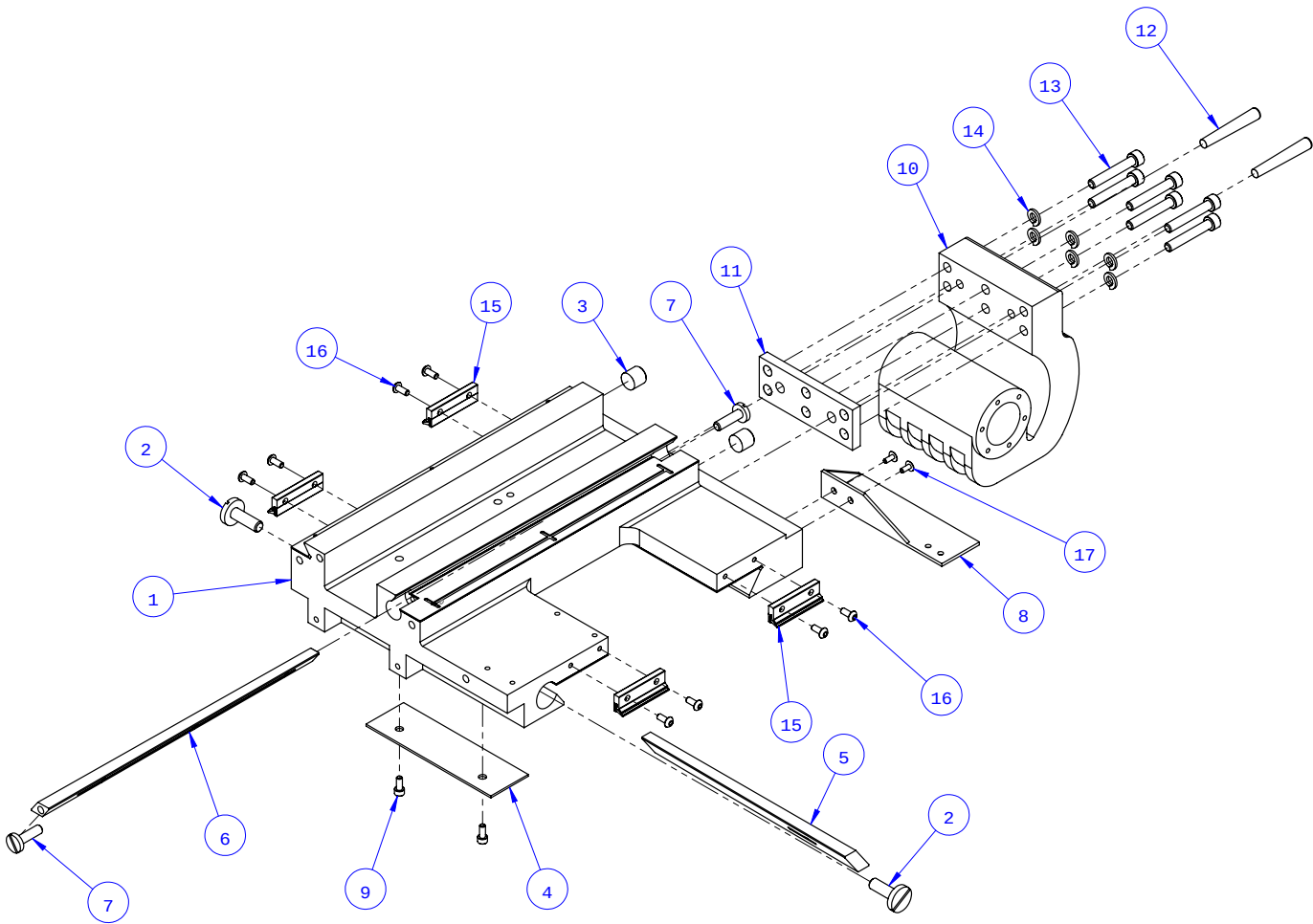
13.3 Bed Casting Assembly

ID	Description	Alternative Part Number	Quantity
1	Lathe Bed Casting, 15LX (PN 34164)	—	1
2	Ball Screw Cover, Z-Axis, 15L (PN 34167)	—	1
3	Belt Cover, Z-Axis, 15L (PN 34185)	—	1
4	Z-Axis Motor Cover, 15L (PN 34192)	—	1
5	Spacer, Z Ball Screw Bearing, 15L (PN 34174)	—	3
6	Bearing Cap, 15L (PN 34198)	—	1
7	Felt Washer, 15L (PN 34199)	—	1
8	Bearing Mount Block Left Hand, 15L (PN 34201)	—	1
9	Bearing Mount Block Right Hand, 15L (PN 34176)	—	1
10	Bearing Cap, 15L (PN 34178)	—	1
11	5M-HTD-60T Pulley, 14 mm Bore, 25 mm Height, 15LX Z-Axis (PN 54958)	—	1
12	Ball Screw Pulley Mounting Flange, Fits 34232 (PN 34234)	—	1
13	Bumper, Z-Axis Right Hand, 15L (PN 34170)	—	1
14	Bumper, Z-Axis Left Hand, 15L (PN 34202)	—	1
15	Z-Axis Motor Mount, 15LX (PN 54366)	—	1
16	Bearing, Z Ball Screw Mount Right Hand, 15L (PN 34177)	—	1
17	Ball Screw Pulley Locking Sleeves, Fits 34232 (PN 34233)	—	2
18	Spanner Nut, M17 × 1, 15L (PN 34179)	—	1
19	Spanner Nut, M15 × 1, 15L (PN 34197)	—	1
20	Bearing, Z Ball Screw Mount Left Hand, 15L (PN 34200)	—	1
21	Seal, Z-Axis Ball Screw, 15L (PN 34172)	—	1
22	Ball Screw Assembly, Z-Axis, 15L (PN 33224)	—	1
23	Screw, Socket Head Cap, M5 × 0.8 - 12 (PN 31353)	McMaster Carr: 91290A228	19
24	Screw, Socket Head Cap, M6 × 1 - 25 (PN 31685)	McMaster-Carr: 91290A330	8
25	Taper Pin, 6 mm × 30 mm, M4 Tapped (PN 34173)	—	4
26	Screw, Button Head Cap, M5 × 0.8 - 10, Stainless Steel (PN 54419)	McMaster Carr: 92095A208	11
27	Screw, Socket Head Cap, M8 × 1.25 - 30 (PN 30544)	McMaster-Carr: 91290A434	3
28	Washer, Flat, M8 (PN 30531)	McMaster-Carr: 91166A270	3
29	Washer, Split Lock, M8 (PN 32339)	—	3

13: DIAGRAMS AND PARTS LISTS

13.4 Z-Axis Saddle Assembly

13.4 Z-AXIS SADDLE ASSEMBLY



13: DIAGRAMS AND PARTS LISTS

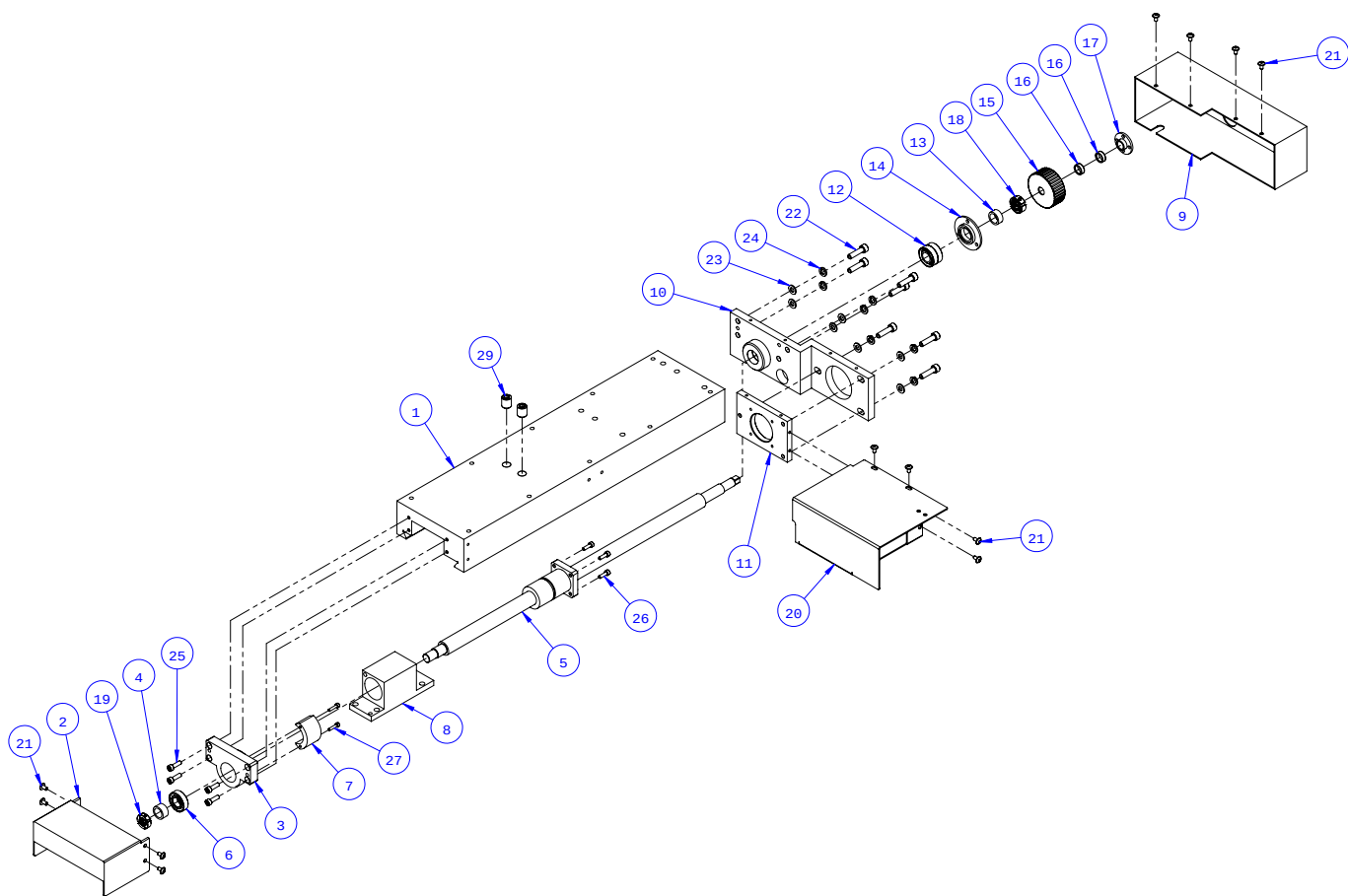
13.4 Z-Axis Saddle Assembly

ID	Description	Alternative Part Number	Quantity
1	Carriage Saddle Casting, 15L (PN 34220)	–	1
2	Gib Screw, Z-Axis, 15L (PN 34218)	–	2
3	Bumper, X-Axis Upper, 15L (PN 34222)	–	2
4	Plate, Carriage Saddle Casting, 15L (PN 34215)	–	1
5	Gib, Z-Axis, 15L (PN 34216)	–	1
6	Gib, X-Axis, 15L (PN 34213)	–	1
7	Gib Screw, X-Axis, 15L (PN 34212)	–	2
8	X-Axis Cable Carrier Mount, 15LX (PN 54390)	–	1
9	Screw, Socket Head Cap, M5 × 0.8 - 12 (PN 31353)	McMaster Carr: 91290A228	2
10	Ball Nut Carrier, Z-Axis, 15L (PN 34194)	–	1
11	Mount Pad, Z Ball Nut Carrier, 15L (PN 34193)	–	1
12	Taper Pin, 8 mm × 65 mm, M5 Tapped (PN 34196)	–	2
13	Screw, Socket Head Cap, M8 × 1.25 - 50 (PN 54387)	McMaster Carr: 91290A448	6
14	Washer, Split Lock, M8 (PN 32339)	–	6
15	15LX Z Way Wiper Seal (PN 34217)	–	4
16	Screw, Button Head Cap, M5 × 0.8 - 12, Stainless Steel (PN 37716)	McMaster-Carr: 92095A210	8
17	Screw, Button Head Cap, Flanged M4 × 0.7 - 10, 18-8 Stainless (PN 50654)	McMaster Carr: 90909A718	2

13: DIAGRAMS AND PARTS LISTS

13.5 X-Axis Cross Slide Assembly

13.5 X-AXIS CROSS SLIDE ASSEMBLY



13: DIAGRAMS AND PARTS LISTS

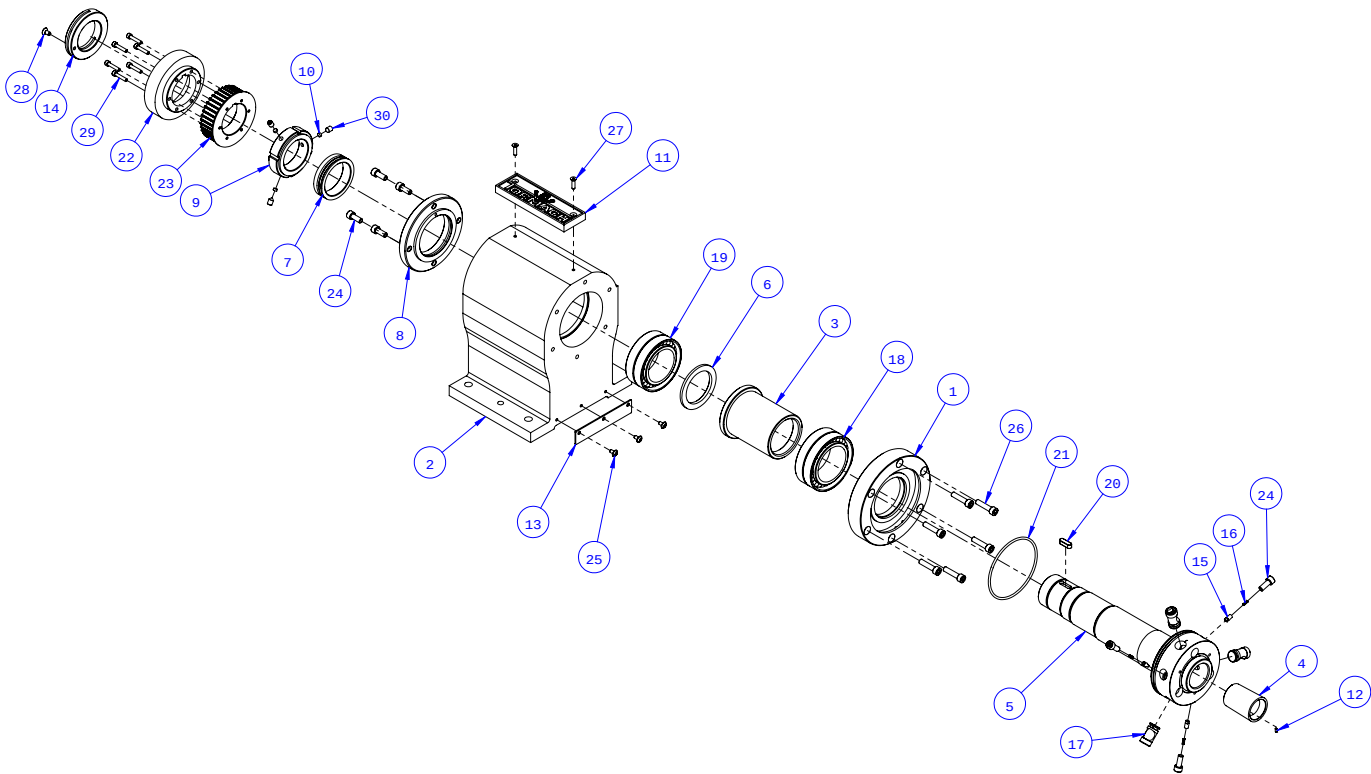
13.5 X-Axis Cross Slide Assembly

ID	Description	Alternative Part Number	Quantity
1	Cross Slide Casting, 15L (PN 34243)	—	1
2	Cover, Z-Axis Cross Slide, Lower, 15LX (PN 54386)	—	1
3	Mounting Plate, X-Axis Ball Screw, Lower, 15L (PN 34208)	—	1
4	Spacer, X-Axis Lower Ball Screw Bearing, 15L (PN 34209)	—	1
5	Ball Screw Assembly, X-Axis, 15L (PN 33223)	—	1
6	Bearing, Z Ball Screw Mount Left Hand, 15L (PN 34200)	—	1
7	Hard Stop, X-Axis, 15LX (PN 54377)	—	1
8	Ball Nut Carrier Block, X-Axis, 15L (PN 34245)	—	1
9	Belt Cover, X-Axis, 15L (PN 34235)	—	1
10	Motor Mount, X-Axis, 15L (PN 34223)	—	1
11	X-Axis Motor Adapter Plate, 15LX (PN 54375)	—	1
12	Bearing, Z Ball Screw Mount Right Hand, 15L (PN 34177)	—	1
13	Spacer, X-Axis Upper Ball Screw Bearing, 15L (PN 34225)	—	1
14	Bearing Cap (PN 34229)	—	1
15	5M-HTD-40T Pulley, 14 mm Bore, 25 mm Height (PN 34232)	—	1
16	Ball Screw Pulley Locking Sleeves, Fits 34232 (PN 34233)	—	2
17	Ball Screw Pulley Mounting Flange, Fits 34232 (PN 34234)	—	1
18	Spanner Nut, M17 × 1, 15L (PN 34179)	—	1
19	Spanner Nut, M15 × 1, 15L (PN 34197)	—	1
20	X-Axis Motor Cover, 15LX (PN 54376)	—	1
21	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	12
22	Screw, Socket Head Cap, M8 × 1.25 - 30 (PN 30544)	McMaster-Carr: 91290A434	7
23	Washer, Flat, M8 (PN 30531)	McMaster-Carr: 91166A270	7
24	Washer, Split Lock, M8 (PN 32339)	—	7
25	Screw, Socket Head Cap, M6 × 1 - 20 (PN 30832)	McMaster-Carr: 91290A326	4
26	Screw, Socket Head Cap, M5 × 0.8 - 16 (PN 30546)	McMaster- Carr: 91290A232	4
27	Screw, Socket Head Cap, M4 × 0.7 - 18 (PN 38734)	McMaster Carr: 91290A164	2
28	Screw, Flat Point Set, M5 × 0.8 - 5 (PN 34221)	McMaster Carr: 93245A213	1
29	Screw, Flat Point Set, M16 × 2 - 20 (PN 55348)	McMaster Carr: 93245A670	2

13: DIAGRAMS AND PARTS LISTS

13.6 Headstock Assembly

13.6 HEADSTOCK ASSEMBLY



13: DIAGRAMS AND PARTS LISTS

13.6 Headstock Assembly

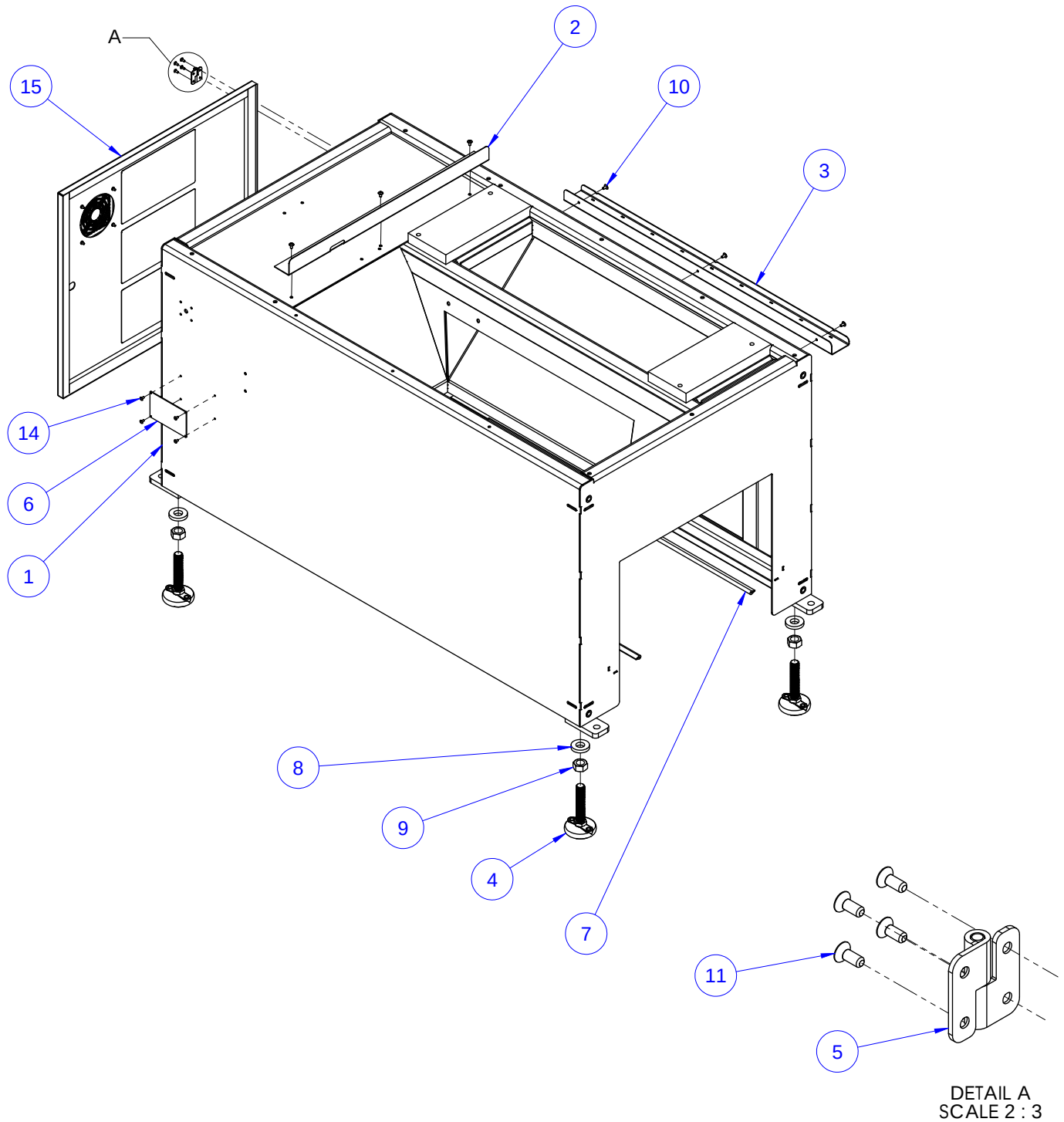
ID	Description	Alternative Part Number	Quantity
1	Nose Cap, Headstock, 15L (PN 34265)	—	1
2	Headstock Casting, 15L (PN 34272)	—	1
3	Spacer Sleeve, Spindle Bearing, 15L (PN 34260)	—	1
4	5C Insert, 15L (PN 33325)	—	1
5	Spindle, 15L (PN 34252)	—	1
6	Spacer Ring, Spindle, 15L (PN 34261)	—	1
7	Spindle Seal Sleeve, 15L (PN 34263)	—	1
8	Headstock Tail Cap, 15L (PN 34274)	—	1
9	Spanner Nut, 15L Spindle (PN 34280)	—	1
10	Brass Locking Pad, 15L (PN 34282)	—	3
11	Logo Casting, 15L (PN 34266)	—	1
12	5C Insert Pin, 15L (PN 34251)	—	1
13	Headstock Bed Cover Plate, 15L (PN 54382)	—	1
14	Spindle Lock Nut, 15L (PN 34285)	—	1
15	Cam Lock Retainer Pin, D1-4, 15L (PN 34255)	—	3
16	Cam Lock Retainer Spring, D1-4, 15L (PN 34254)	—	3
17	Cam Lock Eccentric Pin, D1-4, 15L (PN 34256)	—	3
18	Bearing, Angular Contact, 7013AC/DB, 65 mm × 100 mm × 18 mm Bearing, Spindle Nose, 15L (PN 34259)	—	1
19	Bearing, Angular Contact, 7012AC/DB, 60 mm × 95 mm × 18 mm, Spindle Tail, 15L (PN 34262)	—	1
20	Key, Straight, 8 mm × 7 mm - 25 (PN 34257)	—	1
21	O-Ring Seal, Spindle Nose, 15L (PN 34258)	—	1
22	Spindle Pulley Adapter Flange, 15LX (PN 54383)	—	1
23	8M-HTD-38T Pulley, 56 mm Bore, 30 mm Height, 15LX Spindle Pulley (PN 54316)	—	1
24	Screw, Socket Head Cap, M8 × 1.25 - 20 (PN 31895)	McMaster-Carr: 91290A426	7
25	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	3
26	Screw, Socket Head Cap, M8 × 1.25 - 35 (PN 30529)	McMaster Carr: 91290A438	6
27	Screw, Socket Head Countersunk, M5 × 0.8 - 20, Stainless Steel (PN 54384)	McMaster Carr: 92125A214	2
28	Screw, Socket Head Countersunk, M6 × 1 - 12, Stainless Steel (PN 54385)	McMaster Carr: 92125A236	1

13: DIAGRAMS AND PARTS LISTS

13.6 Headstock Assembly

ID	Description	Alternative Part Number	Quantity
29	Screw, Socket Head Cap, M5 × 0.8 - 25 (PN 30530)	McMaster Carr: 91290A252	6
30	Screw, Cone Point Set, M8 × 12 90° (PN 34281)	—	3

13.7 STAND ASSEMBLY

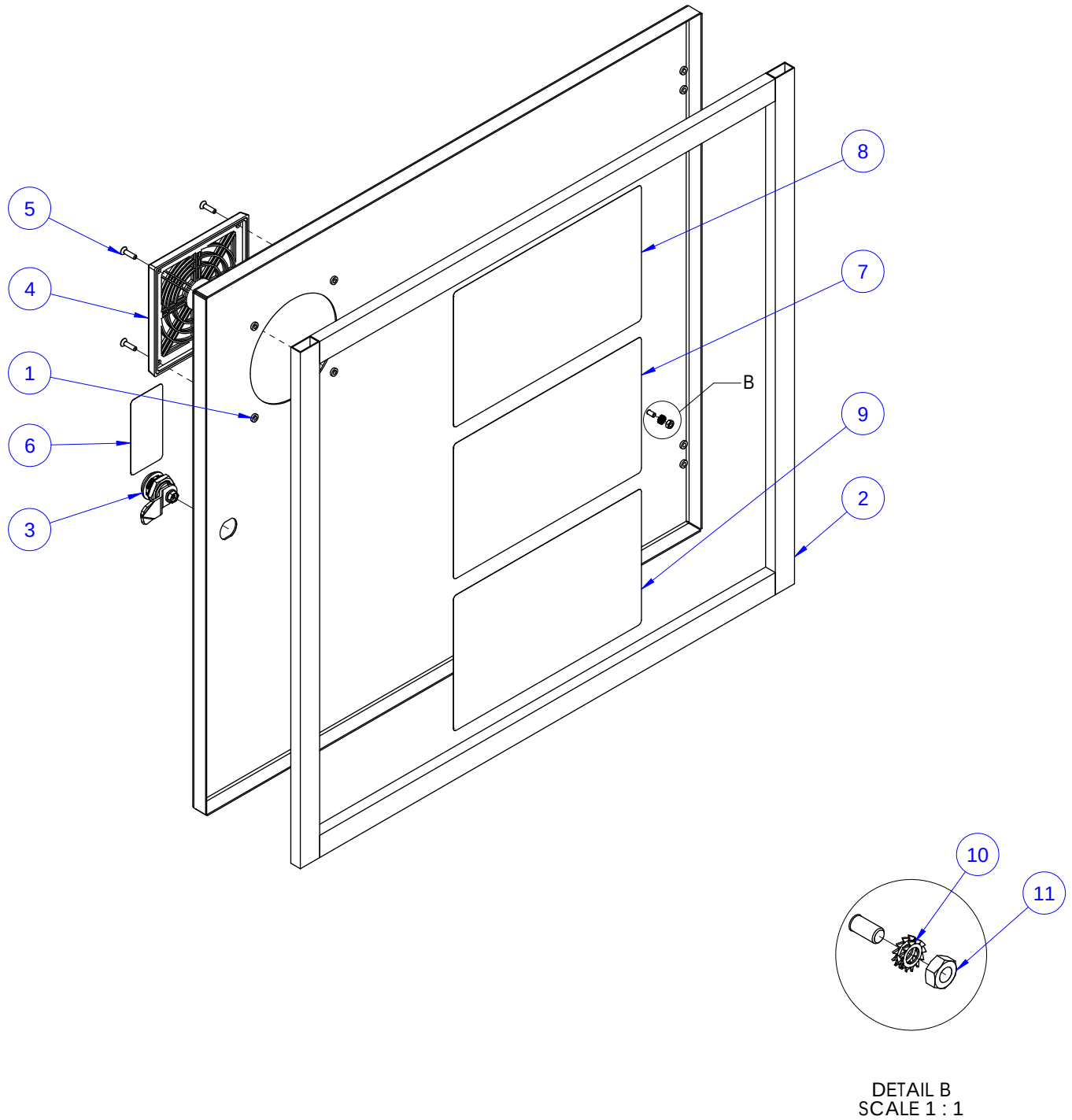


13: DIAGRAMS AND PARTS LISTS

13.7 Stand Assembly

ID	Description	Alternative Part Number	Quantity
1	Stand Core Assembly, 15LX (PN 54493)	—	1
2	Removable Angle for 15LX Stand (PN 54580)	—	1
3	Cable Management Guide, 15LX (PN 54495)	—	1
4	Machine Foot, M20 × 150 mm (PN 54494)	—	4
5	Hinge, Surface-Mount, Lift-Off Style, Left Side, 22 mm Hole Spacing (PN 53479)	—	2
6	Serial Number Steel Blank Plate, 74 mm × 124 mm, 18 ga Stainless Steel (PN 53361)	—	1
7	Rubber Edge Gasket (PN 54573)	—	2
8	Washer, Oversized Flat, M20, 7.0 - 9.0 mm Thickness (PN 52488)	McMaster Carr: 97310A122	4
9	Nut, Hex, M20, Zinc Plated (PN 52618)	McMaster Carr: 90591A230	4
10	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	6
11	Screw, Socket Head Countersunk, M4 × 0.7 - 10, Stainless Steel (PN 52958)	McMaster Carr: 92125A190	8
14	Blind Rivet, 5-32 Diameter, 1-4 Length (PN 52521)	McMaster Carr: 93372A425	4
15	Electrical Cabinet Door Assembly, 15LX (PN 54759)	—	1

13.7.1 Electrical Cabinet Door Assembly



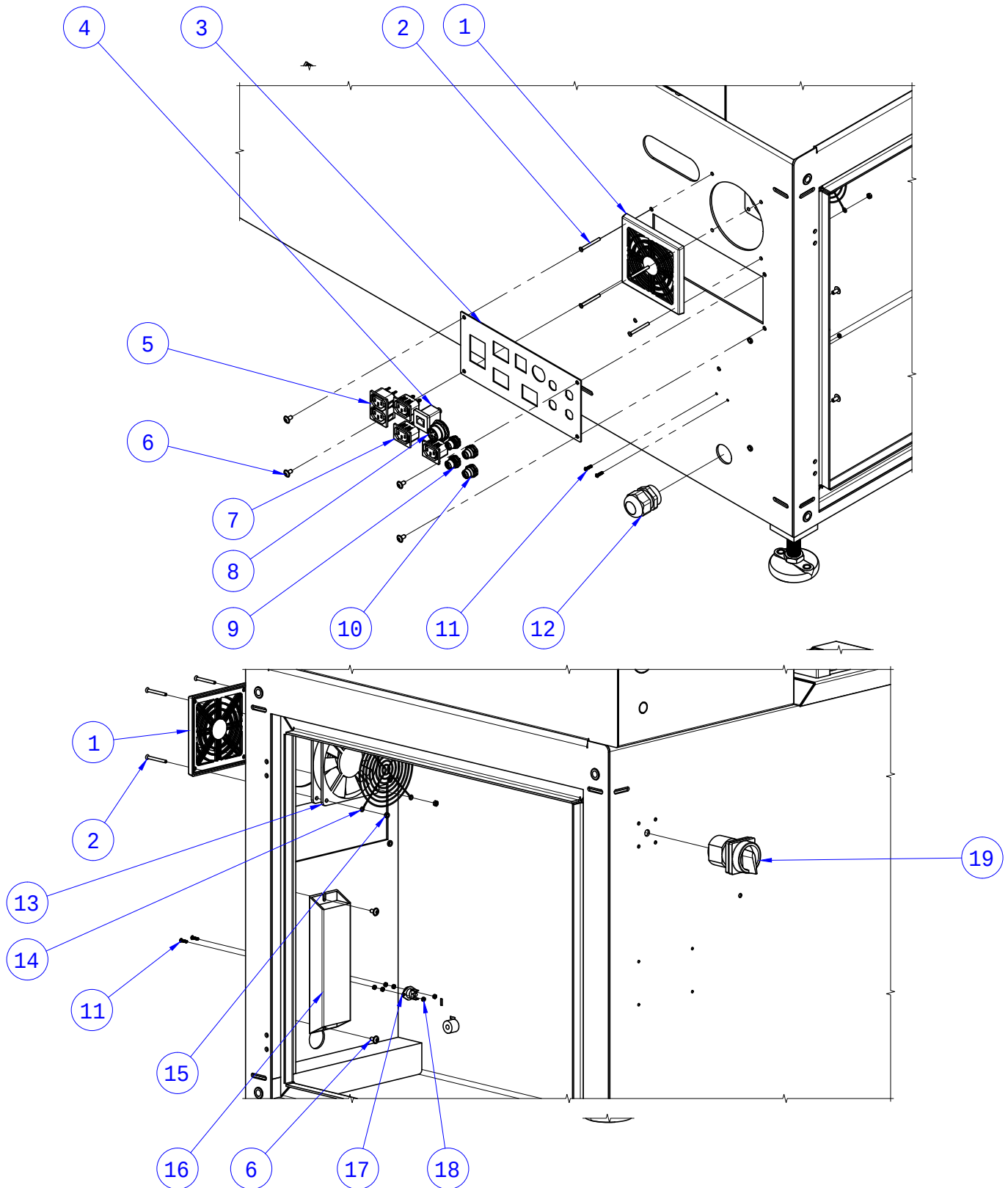
13: DIAGRAMS AND PARTS LISTS

13.7 Stand Assembly

ID	Description	Alternative Part Number	Quantity
1	Electrical Cabinet Door, 15LX (PN 54491)	—	1
2	Rubber Door Gasket, Electrical Cabinet, 1 in. × 0.5 in., 2.4 m Long (PN 51669)	—	1
3	Cam Lock, Zinc Plated, Hex Drive, Quarter-Turn Counterclockwise to Open (PN 51588)	—	1
4	Electrical Cabinet Exhaust Cover Retainer (PN 37947)	McMaster-Carr: 19155K23	1
5	Screw, Socket Head Countersunk, M4 × 0.7 - 16 (PN 52380)	McMaster Carr: 92125A194	4
6	Cabinet Lock Label (PN 53744)	—	1
7	Inner Door I/O Board Connector Layout Label, 15LX (PN 54660)	—	1
8	Inner Door Electrical Cabinet Layout Label, 15LX (PN 54659)	—	1
9	Inner Door Connector Pinout Diagrams Label, 15LX (PN 54661)	—	1
10	Washer, Star Lock, Metric, Stainless Steel (PN 50837)	McMaster Carr: 91120A130	2
11	Nut, Hex, M4, Zinc Plated (PN 38358)	McMaster-Carr: 90591A255	1

13.8 ELECTRICAL CABINET ASSEMBLY

13.8.1 Exterior Layout

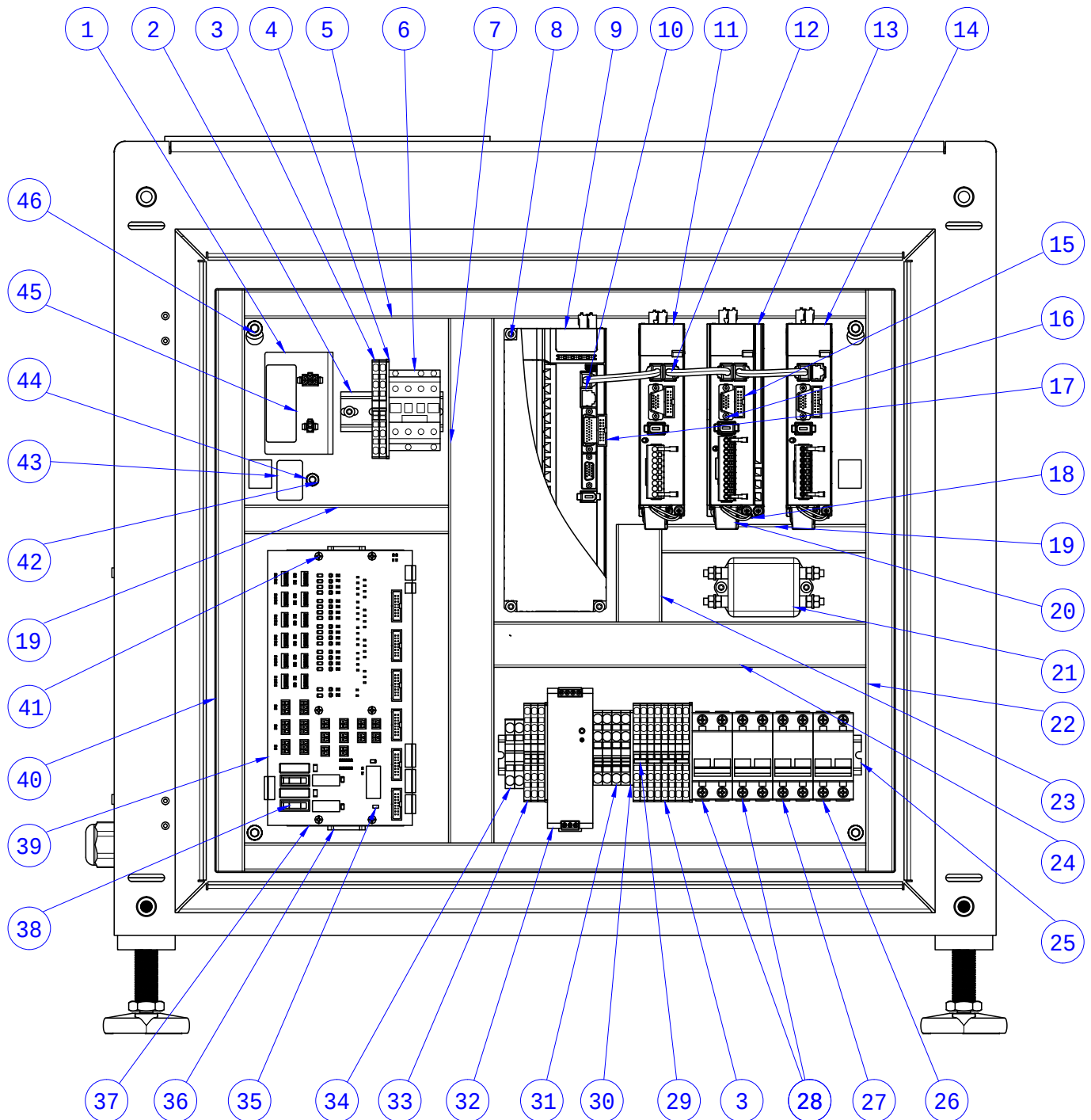


13: DIAGRAMS AND PARTS LISTS

13.8 Electrical Cabinet Assembly

ID	Description	Alternative Part Number	Quantity
1	Electrical Cabinet Exhaust Cover Retainer (PN 37947)	McMaster-Carr: 19155K23	2
2	Screw, Phillips Head Countersunk, M4 × 0.7 - 40, Zinc Plated (PN 51327)	McMaster-Carr: 91420A250	4
3	Rear Connections Panel, 15LX (PN 54409)	—	1
4	RJ45 Bulkhead Connector (Ethernet), 1 m, J12 (PN 53771)	—	1
5	Power Outlet, Double Socket, IEC 320-2-2/F C13, 4.8 mm Spade Terminals, 1.5 mm Panel, 32.5 × 48.3 Cutout (PN 51559)	—	1
6	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	6
7	Power Outlet, Single Socket, IEC 320-2-2/F C13, 4.8 mm Spade Terminals, 32.5 × 25.5 mm Cutout (PN 51560)	—	3
8	Circular Connector, Front-Nut Mount Receptacle, 5 Pin Female, 20 mm Cutout, Round, Solder (PN 51538)	—	1
9	M12 Plug, Front Nut, Panel Mount, Female Pins, 4-Position, A Keying (PN 51543)	—	2
10	Circular Connector, M12, Rear Panel Mount, Plug, 4 Pin Female (PN 51293)	—	2
11	Screw, Button Head Cap, M3 × 0.5 - 12, Stainless Steel (PN 31198)	McMaster Carr: 92095A183	2
12	Cable Cord Grip M32 × 1.5 / 15 mm Nut (PN 54652)	—	1
13	Fan (24 Vdc, 2.5 A, 120 mm) (PN 37526)	McMaster-Carr: 8774N18	1
14	Fan Grill, Finger Guard (105 mm × 105 mm Screw Hooks) (PN 30515)	—	1
15	Nut, Hex, M4, Zinc Plated (PN 38358)	McMaster-Carr: 90591A255	4
16	Brake Resistor Harness, 15LX (PN 54406)	—	1
17	Thermal Switch, Manual Reset, Ceramic, 220 V, 10 A, 85C (PN 37420)	—	1
18	Nut, Hex, M3 x 0.5, Stainless Steel (PN 53327)	McMaster Carr: 90591A250	6
19	Main Disconnect Switch (PN 30454)	—	1

13.8.2 Interior Layout



13: DIAGRAMS AND PARTS LISTS

13.8 Electrical Cabinet Assembly

ID	Description	Alternative Part Number	Quantity
1	Electrical Connector Bulkhead, 15LX (PN 54404)	—	1
2	DIN Rail (35 × 7.5), 90 mm Long (PN 54480)	—	1
3	Terminal Block, 4 Position, Spring Type, Internally Jumpered, JUT3-4/2-2 (PN 51290)	—	10
4	Terminal Block End Plate, JUT3-4/2-2 Series (PN 51291)	—	3
5	Slotted Wire Conduit, 25 × 60 mm, 2 m, PVC (PN 52579)	—	548 mm
6	Contactor Relay, 3-Position, 220 V, 12 A, 24 Vdc Coil, Screw Terminal (PN 37343)	—	1
7	Slotted Wire Conduit, 40 × 60 mm, 2 m, PVC (PN 52580)	—	463 mm
8	Screw, Button Head Cap, M5 × 0.8 - 8 (PN 34636)	McMaster-Carr: 92095A207	38
9	AC Servo Drive, SV670N-INT, Size E, EtherCAT, 220 V, 27 A, STO Function (PN 52546)	—	1
10	Cable, Ethernet, Cat5e, 1 ft (12 in.), Blue (PN 54569)	—	1
11	AC Servo Drive, SV660N, Size A, EtherCAT, 220 V, 2.8 A, STO Function (PN 51194)	—	1
12	Cable, Ethernet, Cat5e, 0.5 ft (6 in.) (PN 51478)	—	1
13	AC Servo Drive, SV660N, Size B, EtherCAT, 220 V, 5.5 A, STO Function (PN 51259)	—	1
14*	AC Servo Drive, SV660N, Size A, EtherCAT, 220 V, 2.8 A, STO Function (PN 51194)	—	1
15	HD15 Male Pin to IDC 2 × 8 Socket Adapter (PN 51623)	—	2
16	Screw, Button Head, #4-40 × 3/8" (PN 51749)	McMaster Carr: 92949A108	6
17	HD26 Male Pin to IDC 2 × 8 Socket Adapter (PN 54575)	—	1
18	Encoder Battery with Leads, 3.6 V (PN 52245)	—	3
19	Slotted Wire Conduit, 25 × 60 mm, 2 m, PVC (PN 52579)	—	180 mm
20	SV660N Encoder Battery Enclosure (PN 51337)	—	3
21	EMI Filter Harness, 15LX (PN 54405)	—	1
22	Slotted Wire Conduit, 25 × 60 mm, 2 m, PVC (PN 52579)	—	513 mm
23	Slotted Wire Conduit, 40 × 60 mm, 2 m, PVC (PN 52580)	—	86 mm
24	Slotted Wire Conduit, 40 × 60 mm, 2 m, PVC (PN 52580)	—	328 mm
25	DIN Rail (35 × 7.5), 320 mm Long (PN 54477)	—	1
26	Circuit Breaker (6 A, 2 Pole) (PN 54472)	—	1
27	Circuit Breaker (10 A, 2 Pole) (PN 37346)	—	1

13: DIAGRAMS AND PARTS LISTS

13.8 Electrical Cabinet Assembly

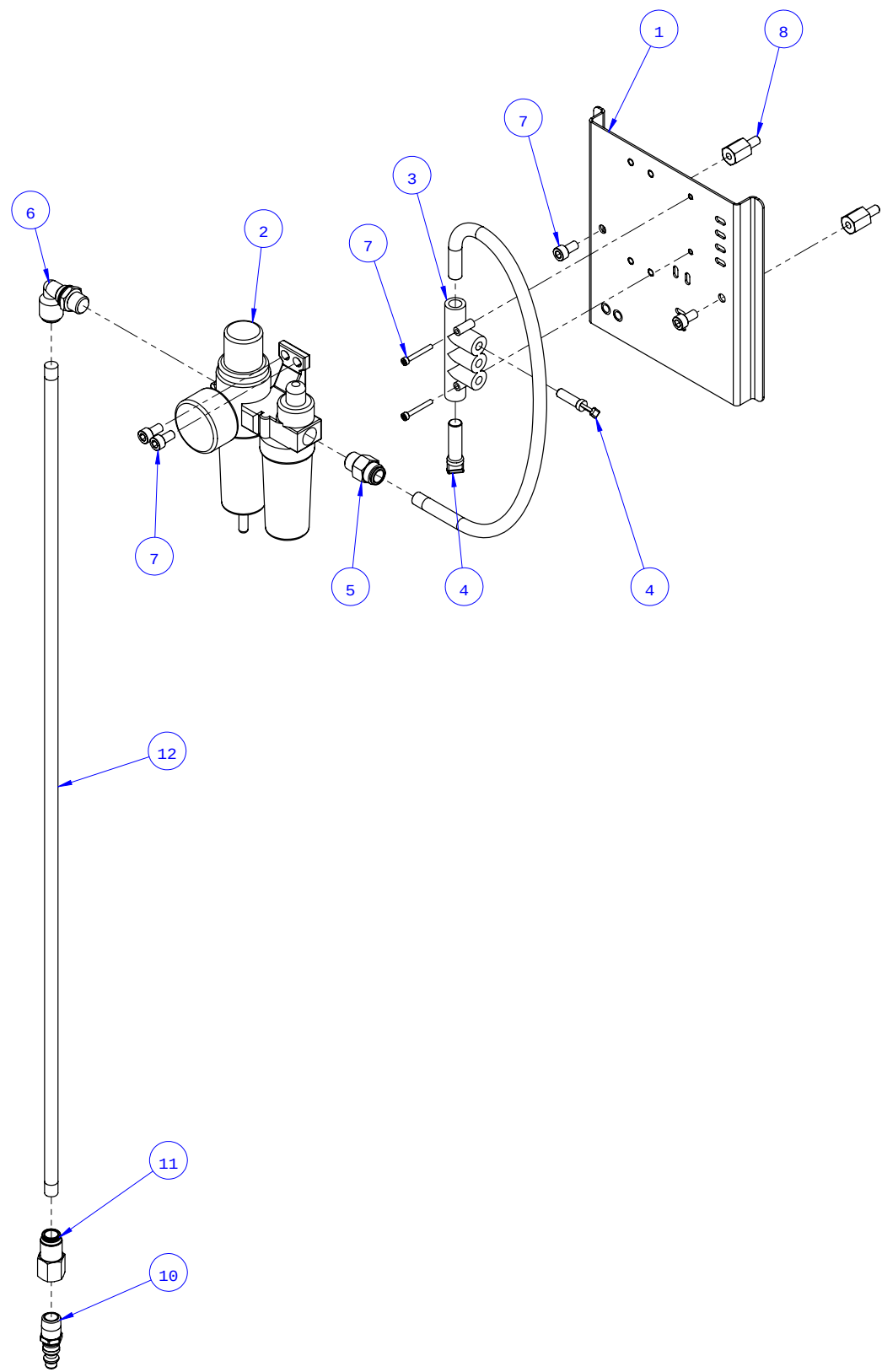
ID	Description	Alternative Part Number	Quantity
28	Circuit Breaker (32 A, 2 Pole) (PN 51576)	—	2
29	Terminal Block Bridge, 2-Position, 6.2 mm Pitch (PN 51292)	—	4
30	Terminal Block End Plate, JUT3-6/1-2 Series (PN 54640)	—	1
31	Terminal Block, 24-8 AWG, 3-Position, Spring Type, Internally Jumpered (PN 54474)	—	4
32	DIN Rail Power Supply, 85-264 VAC, 24 V, 120 W, 5 A, 40 mm × 123.6 mm (PN 54485)	—	1
33	Terminal Block, GND, 4 Position, Spring Type, Internally Jumpered, JUT3-4/2-2PE (PN 51289)	—	3
34	Terminal Block, GND, 24-8AWG, 2-Position, Spring Type, Internally Jumpered (PN 54475)	—	2
35	Jumper, Two Position, Female, 0.100 in. Pitch (PN 52454)	—	1
36	DIN Rail (35 × 7.5), 250 mm Long (PN 54478)	—	1
37	Standoff, 47 mm Hole Spacing, Clip-On Din Rail, Plastic (PN 51473)	—	3
38	<u>Fuse, 3.0 A, 250 V, 5 × 20, Glass, Slow-Blow (PN 31120)</u>	—	2
39	Servo IO Breakout Board (6 Axis) (PN 51252)	—	1
40	Electrical Cabinet Back Plate, 15LX (PN 54403)	—	1
41	Screw, Pan Head, Thread Forming M4 × 1.79 - 10, Zinc Plated (PN 51323)	McMaster-Carr: 90380A376	6
42	Nut, Hex, M6 × 1, Zinc (PN 34896)	McMaster Carr: 90591A151	1
43	Cabinet Ground Label (PN 54716)	—	1
44	Washer, Star Lock, M6, 18-8 Stainless Steel (PN 53809)	McMaster Carr: 91120A160	3
45	Cabinet Bulkhead Label, 15LX (PN 54657)	—	1
46	Screw, Socket Head Cap, M8 × 1.25 - 10 mm, Black Oxide (PN 51884)	McMaster Carr: 91290A414	4

* Optional accessories not included with the base machine.

13: DIAGRAMS AND PARTS LISTS

13.9 Machine Pneumatics Distribution Assembly

13.9 MACHINE PNEUMATICS DISTRIBUTION ASSEMBLY



13: DIAGRAMS AND PARTS LISTS

13.9 Machine Pneumatics Distribution Assembly

ID	Description	Alternative Part Number	Quantity
1	Machine Pneumatics Bracket (PN 51627)	—	1
2	FRL, 1/4 in. NPT Inlet × 1/4 in. NPT Outlet (PN 51635)	—	1
3	Fitting, Reducing Manifold Union, 2x 3/8 in. PTC Ports - 3x 1/4 in. PTC Ports (PN 52316)	—	1
4	Fitting, Plug, 3/8 in. PTC, Plastic (PN 51630)	—	2
5	Fitting, Straight, 1/4 in. NPT (Male) - 3/8 in. PTC (PN 52317)	—	1
6	Fitting, Elbow, 1/4 in. NPT (Male) - 3/8 in. PTC (PN 52318)	—	1
7	Screw, Socket Head Cap, M6 × 1 - 12 (PN 51421)	McMaster-Carr: 91290A318	6
8	Standoff, 16 mm, M6 × 1 - 13 mm Male, M6 × 1 - 10 mm Female, Steel (PN 51566)	McMaster-Carr: 93655A255	2
9	7 Tube, 3/8 in. OD, Polyurethane (Black) (PN 52437)	McMaster-Carr: 5648K261	23 cm
10	Fitting, Brass Coupling, 1/4 Male NPT (PN 31989)	McMaster Carr: 1077T17	1
11	Fitting, Straight, 1/4 in. NPT (Female) - 3/8 in. PTC (PN 52319)	—	1
12	7 Tube, 3/8 in. OD, Polyurethane (Black) (PN 52437)	McMaster-Carr: 5648K261	1

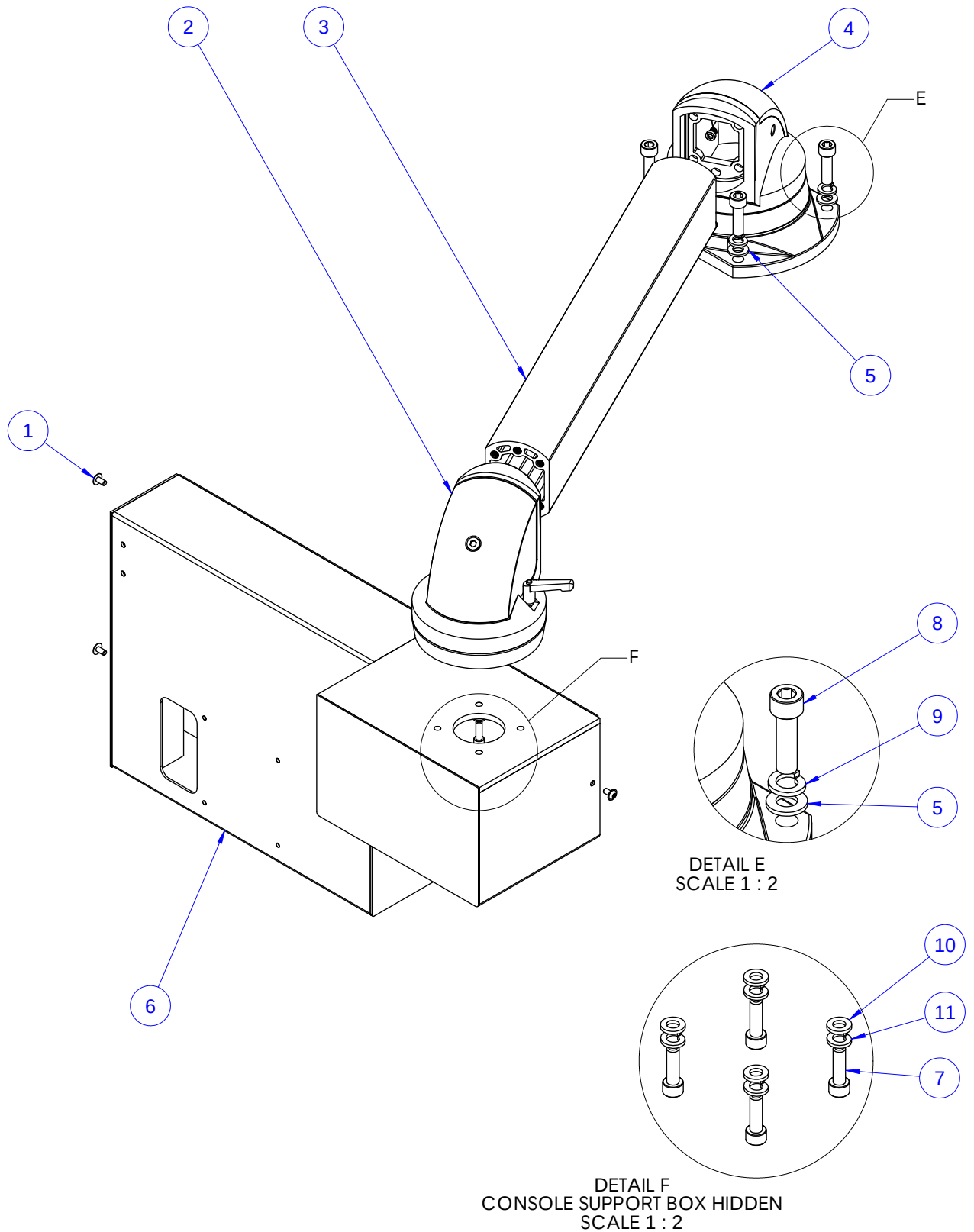
13: DIAGRAMS AND PARTS LISTS

13.10 Enclosure Assembly

13.10 ENCLOSURE ASSEMBLY

13.10.1 Pivot Arm for Console Assembly.....	233
13.10.2 Enclosure Left Side Assembly.....	235
13.10.3 Enclosure Right Side Assembly.....	237
13.10.4 Enclosure Door Assembly.....	240

13.10.1 Pivot Arm for Console Assembly

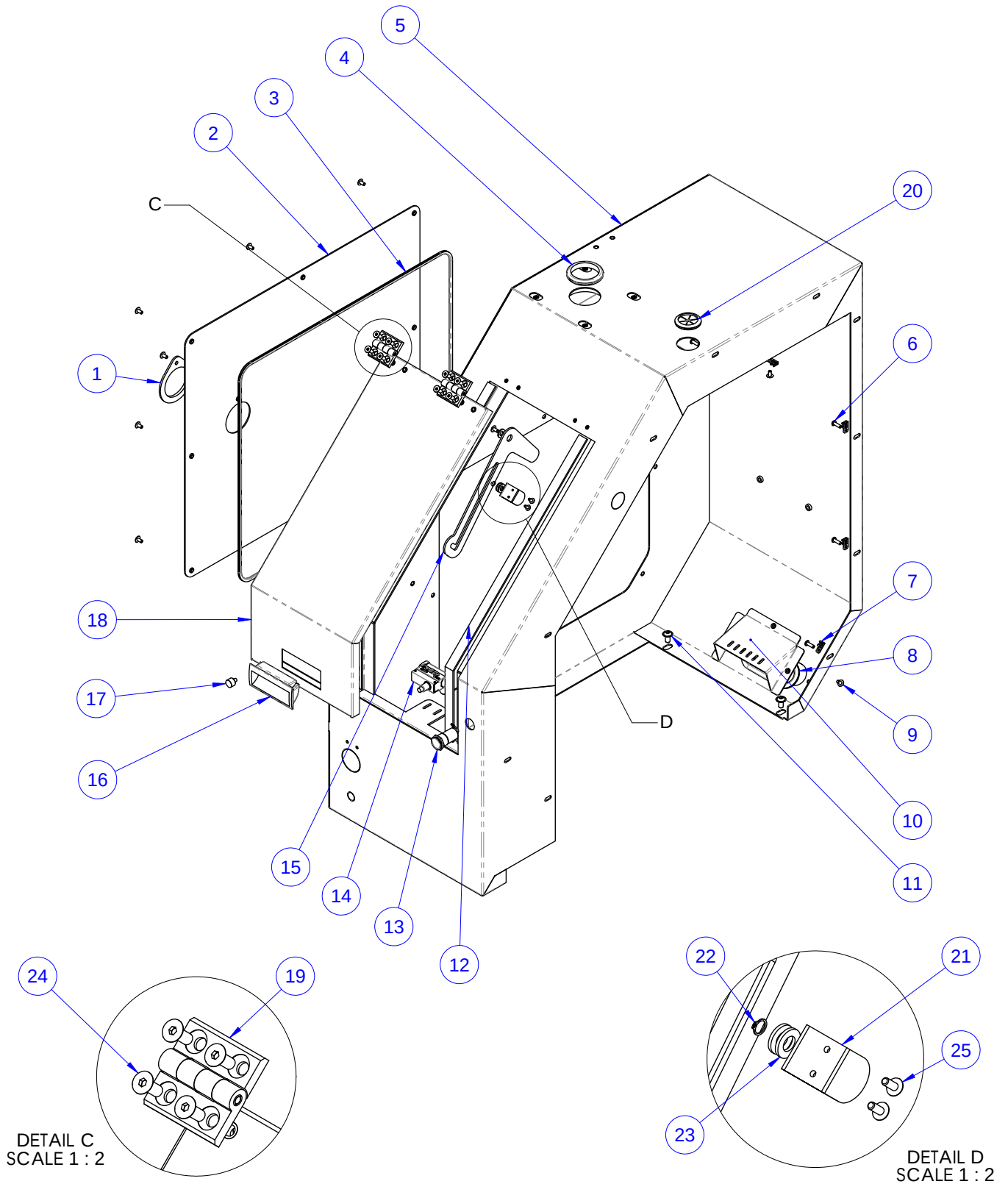


13: DIAGRAMS AND PARTS LISTS

13.10 Enclosure Assembly

ID	Description	Alternative Part Number	Quantity
1	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	4
2	90 Degree Angle Mount for Suspension Arm (PN 54557)	—	1
3	Long Tube for Suspension Arm (PN 54558)	—	1
4	Bench Top Base For Suspension Arm (PN 54559)	—	1
5	Washer, Flat, M10 (PN 30919)	—	4
6	Console Support Box (PN 54560)	—	1
7	Screw, Socket Head Cap, M6 × 1 - 20 (PN 30832)	McMaster-Carr: 91290A326	4
8	Screw, Socket Head Cap, M10 × 1.5 - 35 (PN 31592)	—	4
9	Washer, Split Lock, M10 (PN 31684)	—	4
10	Washer, Flat, M6 (PN 31676)	McMaster-Carr: 91455A120	4
11	Washer, Split Lock, M6 (PN 31379)	—	4

13.10.2 Enclosure Left Side Assembly

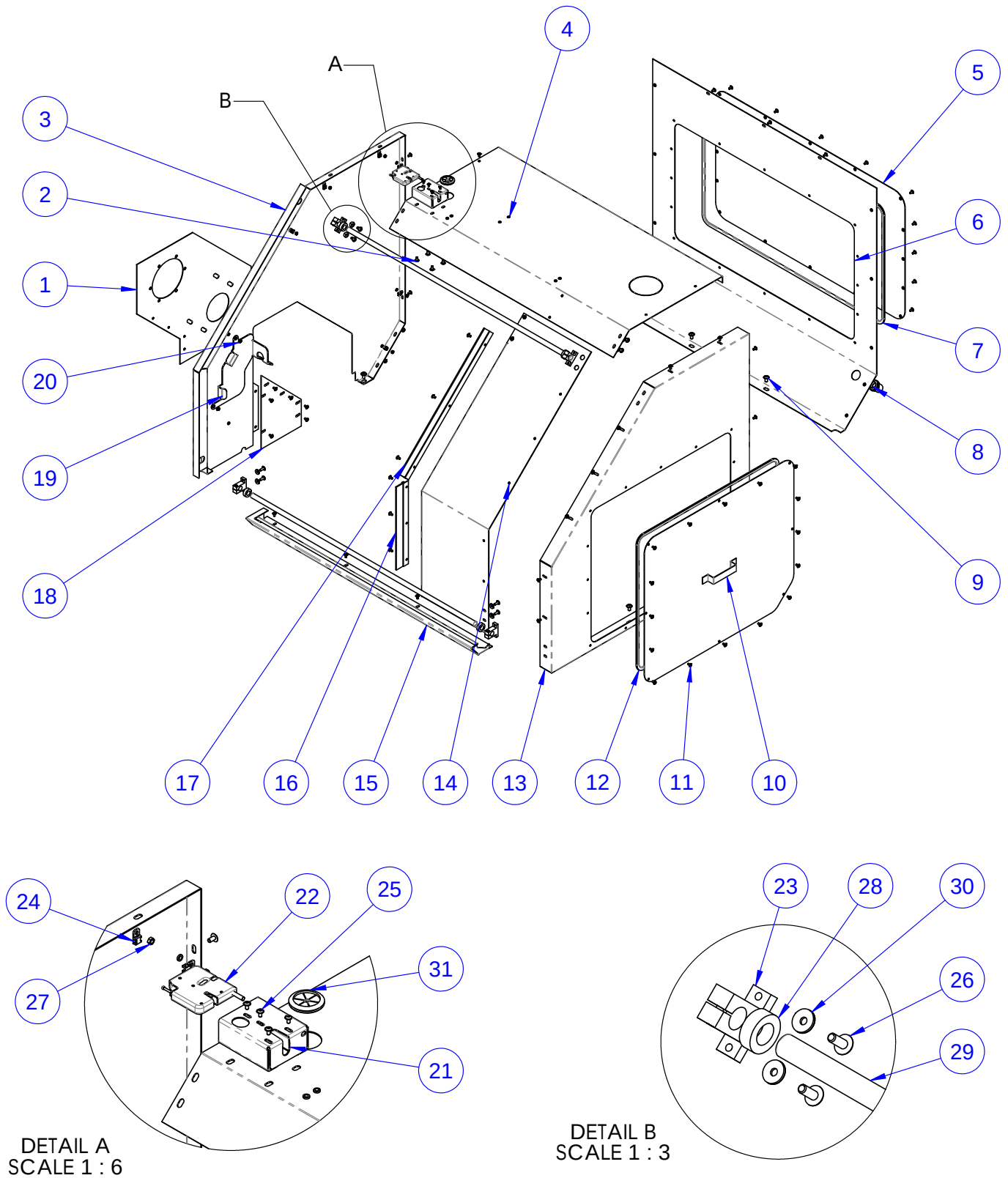


13: DIAGRAMS AND PARTS LISTS

13.10 Enclosure Assembly

ID	Description	Alternative Part Number	Quantity
1	Bar Stock Access Cover, 15L (PN 54608)	—	1
2	Left Plate Window, 15LX (PN 54513)	—	1
3	Left Side Window Trim (1.91 m), 15LX (PN 54519)	—	1
4	Hole Grommet, 50 mm × 55 mm × 3 mm, Rubber (PN 54689)	—	2
5	Left Side Main Cover Panel, 15LX (PN 54511)	—	1
6	Screw, Button Head Cap, M5 × 0.8 - 16, Stainless Steel (PN 51141)	—	9
7	Cable Tie Anchor, Screw Mount, M5 (PN 31460)	—	13
8	Cable Pass-Through Rubber Edge Cover, 15LX (PN 54521)	—	1
9	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	17
10	Cable Pass-Through Doghouse, 15LX (PN 54514)	—	1
11	Screw, Button Head Cap, Flanged M8 × 1.25 - 16, 18-8 Stainless (PN 38468)	McMaster-Carr: 90909A479	4
12	Tesamoll Tape (0.15 m), 15LX (PN 54520)	—	2
13	Push Button, 1NO1NC, White LED (PN 51239)	—	1
14	Limit Switch for Front Door, 15LX (PN 54548)	—	1
15	Hard Stop for 5C Front Door, 15LX (PN 54516)	—	1
16	Door Handle for Front Door, 15LX (PN 54522)	—	1
17	Push-In Bumper, 0.25 in. Stem, 0.38 in. Height (PN 52210)	McMaster-Carr: 9309K68	1
18	Front Door 5C Access, 15LX (PN 54512)	—	1
19	Hinge for Front Door, 15LX (PN 54523)	—	4
20	Hole Grommet, 35 mm × 40 mm × 3 mm, Rubber (PN 55001)	—	1
21	Door Lock Driver Shaft, 15LX (PN 54517)	—	1
22	External Retaining Ring for 5/16 in. OD (PN 54525)	—	1
23	Washer, Flat, M8 (PN 30531)	McMaster-Carr: 91166A270	3
24	Screw, Socket Head Countersunk, M6 × 1 - 16 (PN 54524)	McMaster Carr: 92125A238	8
25	Screw, Button Head Cap, Flanged M4 × 0.7 - 10, 18-8 Stainless (PN 50654)	—	2

13.10.3 Enclosure Right Side Assembly



13: DIAGRAMS AND PARTS LISTS

13.10 Enclosure Assembly

ID	Description	Alternative Part Number	Quantity
1	Spindle Upper Back Panel, 15LX (PN 54526)	—	1
2	Screw, Button Head Cap (Flanged), M6 × 1 - 12, Stainless Steel (PN 38206)	McMaster-Carr: 97654A380	4
3	Enclosure Medium Panel, 15LX (PN 54530)	—	1
4	Enclosure Right Top Panel, 15LX (PN 54536)	—	1
5	Rear Plate Window, 15LX (PN 54537)	—	1
6	Back Panel Cover, 15LX (PN 54533)	—	1
7	Rear Side Window Trim (2.53 m), 15LX (PN 54545)	—	1
8	Fitting, Tank Bulkhead, 3/4 in. GHT (Male) - 1/2 in. NPT (Female) (PN 51508)	—	1
9	Screw, Button Head Cap, Flanged M8 × 1.25 - 16, 18-8 Stainless (PN 38468)	McMaster-Carr: 90909A479	6
10	Right Plate Window, 15LX (PN 54538)	—	1
11	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	71
12	Right Side Window Trim (2.27m), 15LX (PN 54546)	—	1
13	Right Side Enclosure Panel, 15LX (PN 54534)	—	1
14	Enclosure Front Right Panel, 15LX (PN 54539)	—	1
15	Front Drip Catcher Sheet, 15LX (PN 54540)	—	1
16	Short Brush, 15LX (PN 54542)	—	1
17	Long Brush, 15LX (PN 54543)	—	1
18	Spindle Lower Back Panel, 15LX (PN 54541)	—	1
19	Hose Nozzle Bracket, 15LX (PN 54668)	—	1
20	Nut, Flange, M5 × 0.8, Zinc Plated (PN 50577)	—	2
21	Locking Door Switch Bracket (PN 51369)	—	1
22	Locking Door Switch (PN 51328)	—	1
23	Linear Rail Support, Round, 16 mm (PN 37622)	—	4
24	Cable Tie Anchor, Screw Mount, M5 (PN 31460)	—	7
25	Screw, Button Head Cap, Flanged M4 × 0.7 - 8, 18-8 Stainless (PN 51331)	McMaster-Carr: 97654A372	4
26	Screw, Button Head Cap, Flanged M6 × 1 - 16, 18-8 Stainless (PN 54618)	McMaster Carr: 90909A728	8
27	Nut, Hex, M5 × 0.8 (PN 31201)	—	7

13: DIAGRAMS AND PARTS LISTS

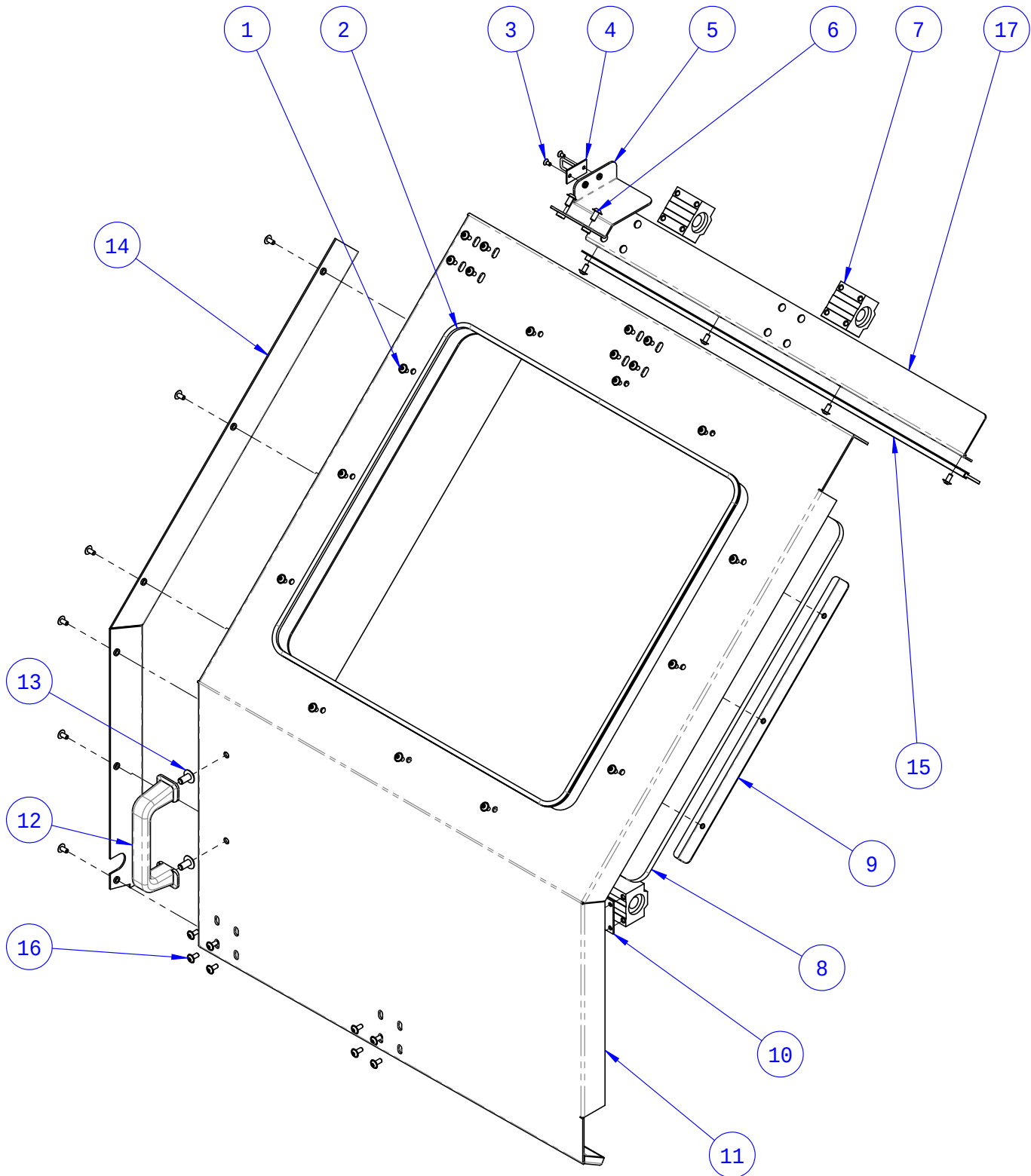
13.10 Enclosure Assembly

ID	Description	Alternative Part Number	Quantity
28	Door Bumper (PN 38357)	—	4
29	Linear Rail, 16 mm Round, 1013 mm, 15LX (PN 54544)	—	2
30	Bonded Neoprene Washer, 1/4 × 5/8 in. OD (PN 53375)	—	8
31	Hole Grommet, 35 mm × 40 mm × 3 mm, Rubber (PN 55001)	—	1

13: DIAGRAMS AND PARTS LISTS

13.10 Enclosure Assembly

13.10.4 Enclosure Door Assembly



13: DIAGRAMS AND PARTS LISTS

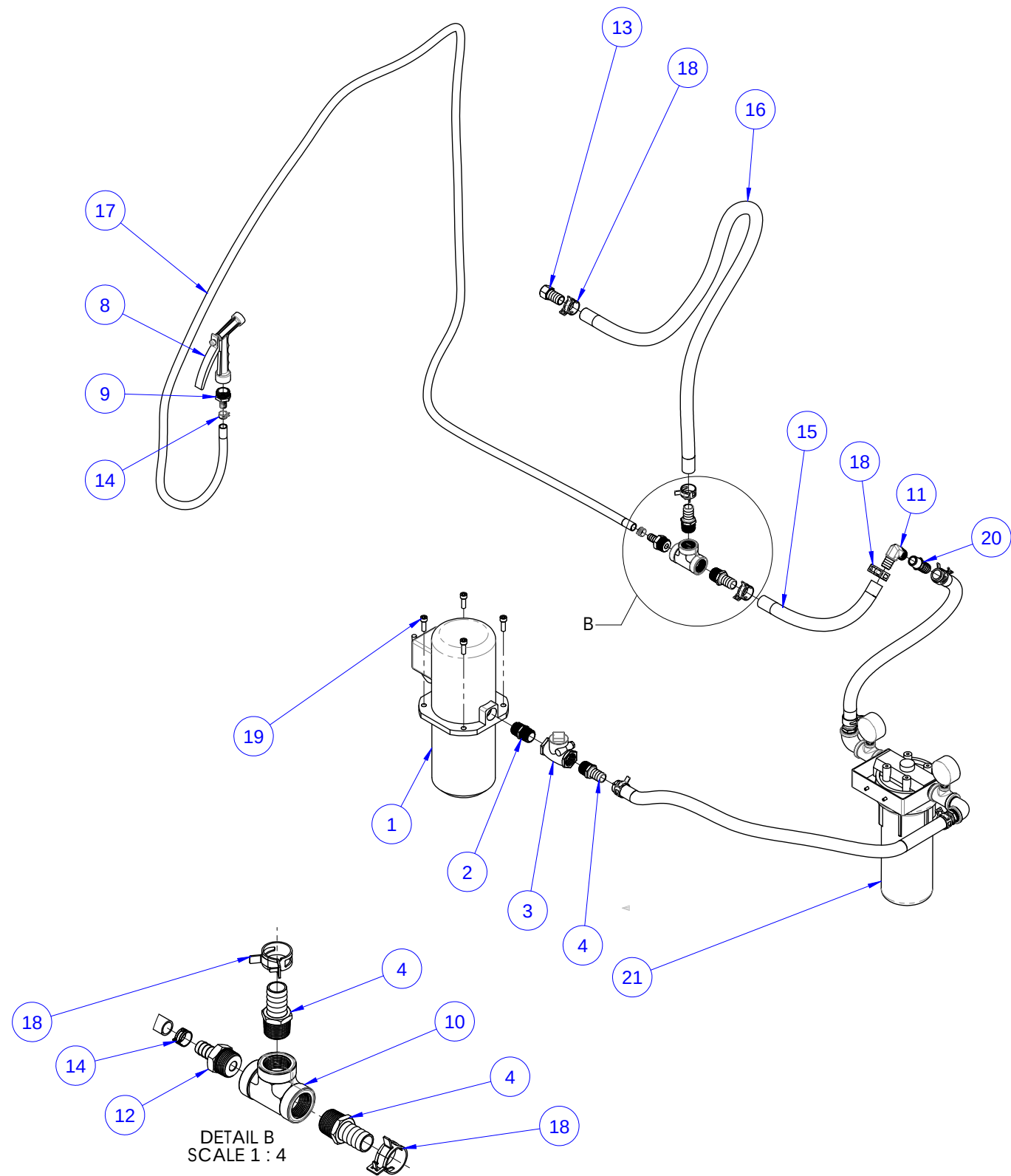
13.10 Enclosure Assembly

ID	Description	Alternative Part Number	Quantity
1	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	12
2	Window Rubber (PN 50512)	—	1
3	Screw, Socket Head Countersunk, M4 × 0.7 - 8, Stainless Steel (PN 51672)	McMaster-Carr: 92125A188	2
4	Locking Door Switch Latch Plate (PN 51329)	—	1
5	Latch Plate, 15LX (PN 54550)	—	1
6	Screw, Button Head Cap (Flanged), M6 × 1 - 12, Stainless Steel (PN 38206)	McMaster-Carr: 97654A380	2
7	Linear Bearing (PN 37610)	—	4
8	Door Window, 15LX (PN 54553)	—	1
9	Enclosure Window Retainer, 15LX (PN 54551)	—	4
10	Bearing Spacer Plate, 15LX (PN 54552)	—	4
11	Door Panel, 15LX (PN 54549)	—	1
12	Grab Handle, 161 mm OAL × 5 mm, 15LX (PN 54555)	—	1
13	Screw, Button Head Cap, Flanged M8 × 1.25 - 16, 18-8 Stainless (PN 38468)	McMaster-Carr: 90909A479	2
14	Inner Door Flange, 15LX Enclosure (PN 54923)	—	1
15	15LX Upper Enclosure Door Brush (PN 55051)	—	1
16	Screw, Button Head Cap (Flanged), M5 × 0.8 - 12 (PN 38890)	McMaster-Carr: 97654A377	16
17	Bracket, 15LX Upper Enclosure Door Brush (PN 55052)	—	1

13: DIAGRAMS AND PARTS LISTS

13.11 Flood Coolant Kit Assembly

13.11 FLOOD COOLANT KIT ASSEMBLY



13: DIAGRAMS AND PARTS LISTS

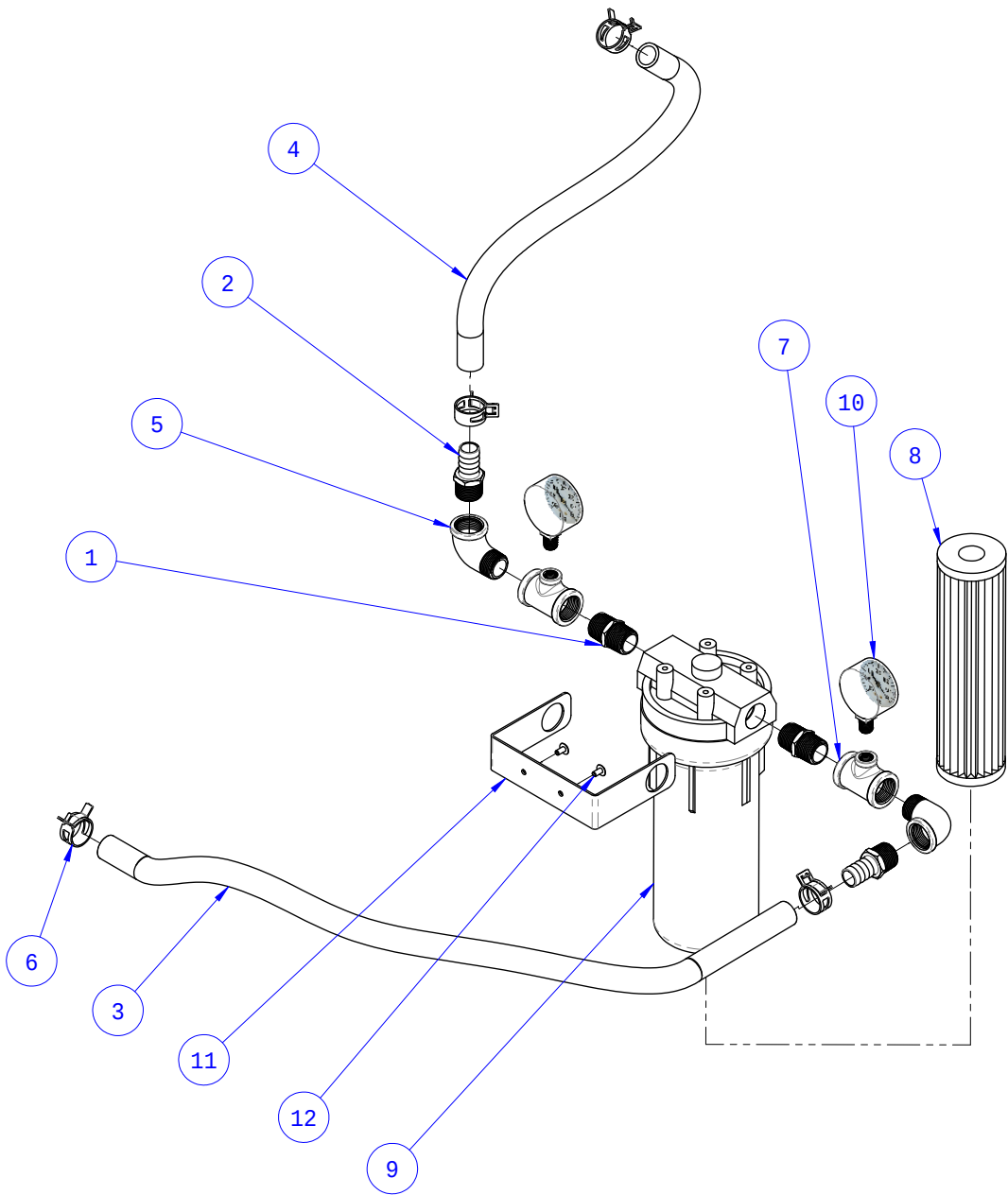
13.11 Flood Coolant Kit Assembly

ID	Description	Alternative Part Number	Quantity
1	Coolant Pump, Dual Voltage (Preset to 240), 1/4 hp, 180 mm Length, 3/4 NPT Outlet, 2 m 14 AWG Cord (with Right-Angle C14 Plug) (PN 51384)	—	1
2	Fitting, Straight Connector, 3/4 in. NPT (Male) - 3/4 in. NPT (Male), Brass (PN 51554)	McMaster Carr: 5485K25	1
3	Fitting, Swing Check Valve, 3/4 in. NPT (Female) - 3/4 in. NPT (Female) (PN 51555)	McMaster Carr: 4708K54	1
4	Fitting, Straight Adapter, 3/4 in. NPT (Male) - 3/4 in. Barb, Brass (PN 51556)	McMaster Carr: 5346K28	3
8	Hose Nozzle, 3/4 in. GHT (PN 51513)	—	1
9	Fitting, Straight, 3/4 in. GHT (Male) - 1/2 in. Barbed, Brass (PN 52411)	McMaster-Carr: 2706N22	1
10	Fitting, Inline Tee 3x 3/4 in. NPT (Female), Brass (PN 54665)	McMaster Carr: 9171K97	1
11	Fitting, Elbow Adapter, 1/2 in. NPT (Male) – 3/4 in. Barb (PN 54664)	McMaster Carr: 53525K32	1
12	Fitting, Straight Adapter, 3/4 in. NPT (Male) – 1/2 in. Barb, Brass (PN 54666)	McMaster Carr: 4374N503	1
13	Fitting, Straight Adapter, 1/2 in. NPT (Female) - 3/4 in. Barb, Brass (PN 54667)	McMaster Carr: 5346K64	1
14	Hose Clamp, Spring, for 5/8 in. OD Hose (PN 51679)	McMaster Carr: 7329K42	2
15	Hose, 3/4 in. ID × 1 in. OD, PVC (PN 51608)	McMaster-Carr: 5233K71	0.43 m
16	Hose, 3/4 in. ID × 1 in. OD, PVC (PN 51608)	McMaster-Carr: 5233K71	1.25 m
17	Hose, Washdown, 7/8 in. OD × 5/8 in. ID - 5 ft, 3/4 in. GHT Fittings (PN 51510)	McMaster Carr: 7110T18	3 m
18	Hose Clamp, Spring, for 1-1/16 in. OD Hose (PN 51558)	McMaster Carr: 7329K16	4
19	Screw, Socket Head Cap, M8 × 1.25 - 25 (PN 34635)	McMaster-Carr: 91290A432	4
20	Fitting, Adapter, 1/2 in. NPT (Male) - 3/4 in. Barb (PN 51552)	McMaster-Carr: 5346K68	1
21	Flood Coolant Filter Assembly, 15LX (PN 54968)	—	1

13: DIAGRAMS AND PARTS LISTS

13.11 Flood Coolant Kit Assembly

Flood Coolant Filter Assembly



13: DIAGRAMS AND PARTS LISTS

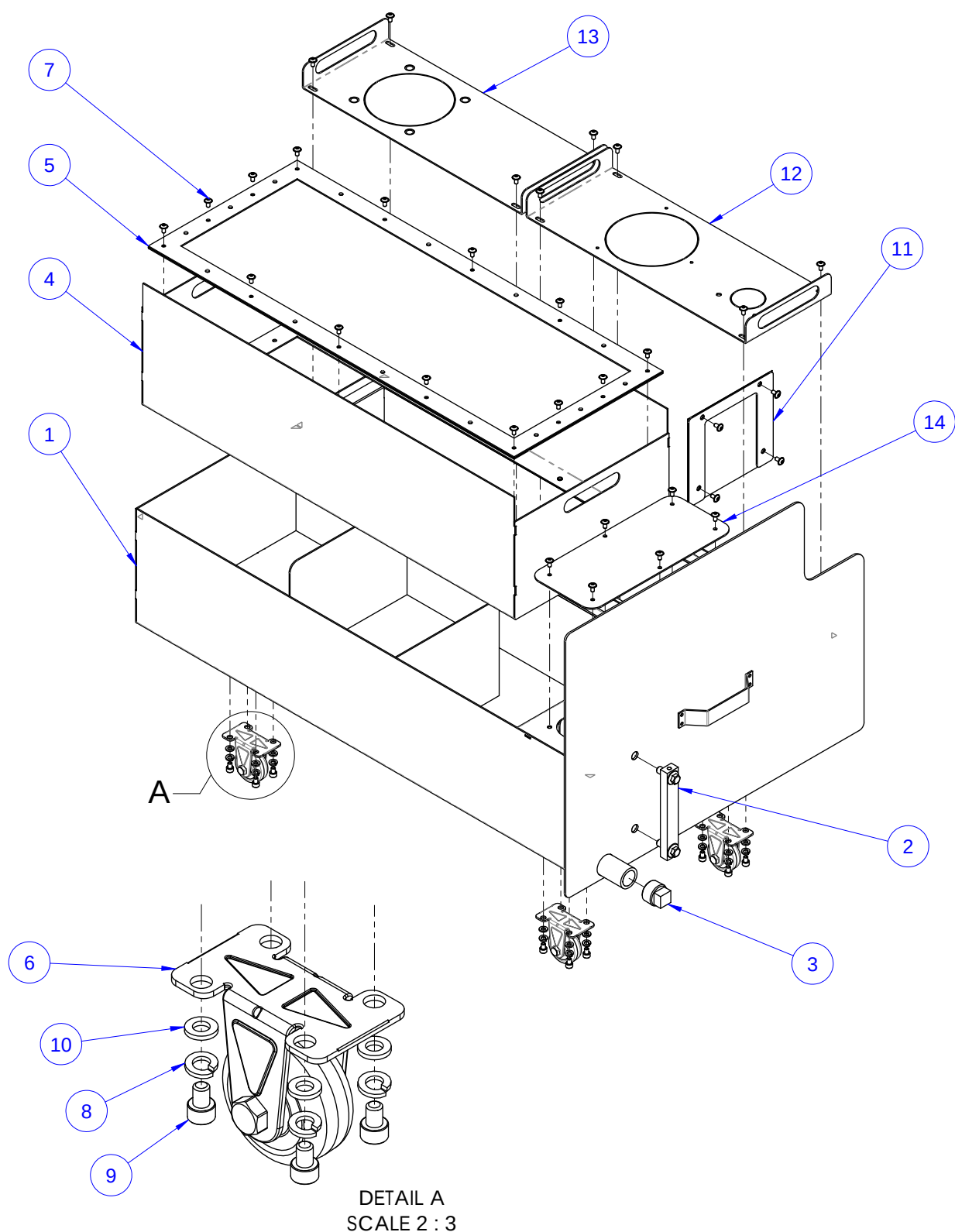
13.11 Flood Coolant Kit Assembly

ID	Description	Alternative Part Number	Quantity
1	Fitting, Straight Connector, 3/4 in. NPT (Male) - 3/4 in. NPT (Male), Brass (PN 51554)	McMaster Carr: 5485K25	2
2	Fitting, Straight Adapter, 3/4 in. NPT (Male) - 3/4 in. Barb, Brass (PN 51556)	McMaster Carr: 5346K28	2
3	Hose, 3/4 in. ID × 1 in. OD, PVC (PN 51608)	McMaster-Carr: 5233K71	0.85 m
4	Hose, 3/4 in. ID × 1 in. OD, PVC (PN 51608)	McMaster-Carr: 5233K71	0.55 m
5	Fitting, Street Elbow Adapter, 3/4 in. NPT (Male) - 3/4 in. NPT (Female), Brass (PN 51606)	McMaster Carr: 4429K154	2
6	Hose Clamp, Spring, for 1 in. OD Hose (PN 51558)	McMaster Carr: 7329K16	4
7	Fitting, Inline Tee Reducer, 2x 3/4 in. NPT (Female) x 1/4 in. NPT (Female), Brass (PN 51610)	McMaster-Carr: 4429K229	2
8	Replacement Filter Cartridge, For Flood Coolant Filter (PN 51615)	McMaster-Carr: 7191K11	1
9	Flood Coolant Filter Housing, 3/4 in. NPT Inlets (PN 51613)	McMaster-Carr: 4422K3	1
10	Fitting, Pressure Gauge, 1/4 in. NPT, 0-60 psi, for Water (PN 51611)	McMaster-Carr: 4089K61	2
11	Upper Bracket, Flood Coolant Filter (PN 52615)	—	1
12	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 90909A722	2

13: DIAGRAMS AND PARTS LISTS

13.11 Flood Coolant Kit Assembly

Coolant Tank Assembly



13: DIAGRAMS AND PARTS LISTS

13.11 Flood Coolant Kit Assembly

ID	Description	Alternative Part Number	Quantity
	Coolant Tank Sheet Metal Assembly - 15LX (PN 54476)	—	1
	Replacement Level Gauge Sight Glass (PN 50868)	—	1
	Fitting, Plug, 1 in. NPT, External Square Drive (PN 52561)	McMaster Carr: 44605K663	1
	Chip Basket, 15LX Coolant Tank (PN 54479)	—	1
	Removable Chip Screen for Chip Basket, 15LX (PN 54481)	—	1
	Coolant Tank Caster, Fixed (PN 54415)	—	4
	Screw, Button Head Cap (Flanged), M5 × 0.8 - 10, Stainless Steel (PN 38205)	McMaster-Carr: 97654A376	32
	Washer, Split Lock, M6 (PN 31379)	McMaster Carr: 92148A180	16
	Screw, Socket Head Cap, M6 × 1 - 8 (PN 51880)	McMaster Carr: 91290A312	16
	Washer, Flat, M6 (PN 31676)	McMaster-Carr: 91455A120	16
	Removable Chip Screen for Coolant Pump 15LX (PN 54488)	—	1
	Oil Skimmer Cover Panel - 15LX (PN 54486)	—	1
	Coolant Pump Cover Panel - 15LX (PN 54484)	—	1
	Front Access Panel - 15LX Coolant Tank(PN 55193)	—	1

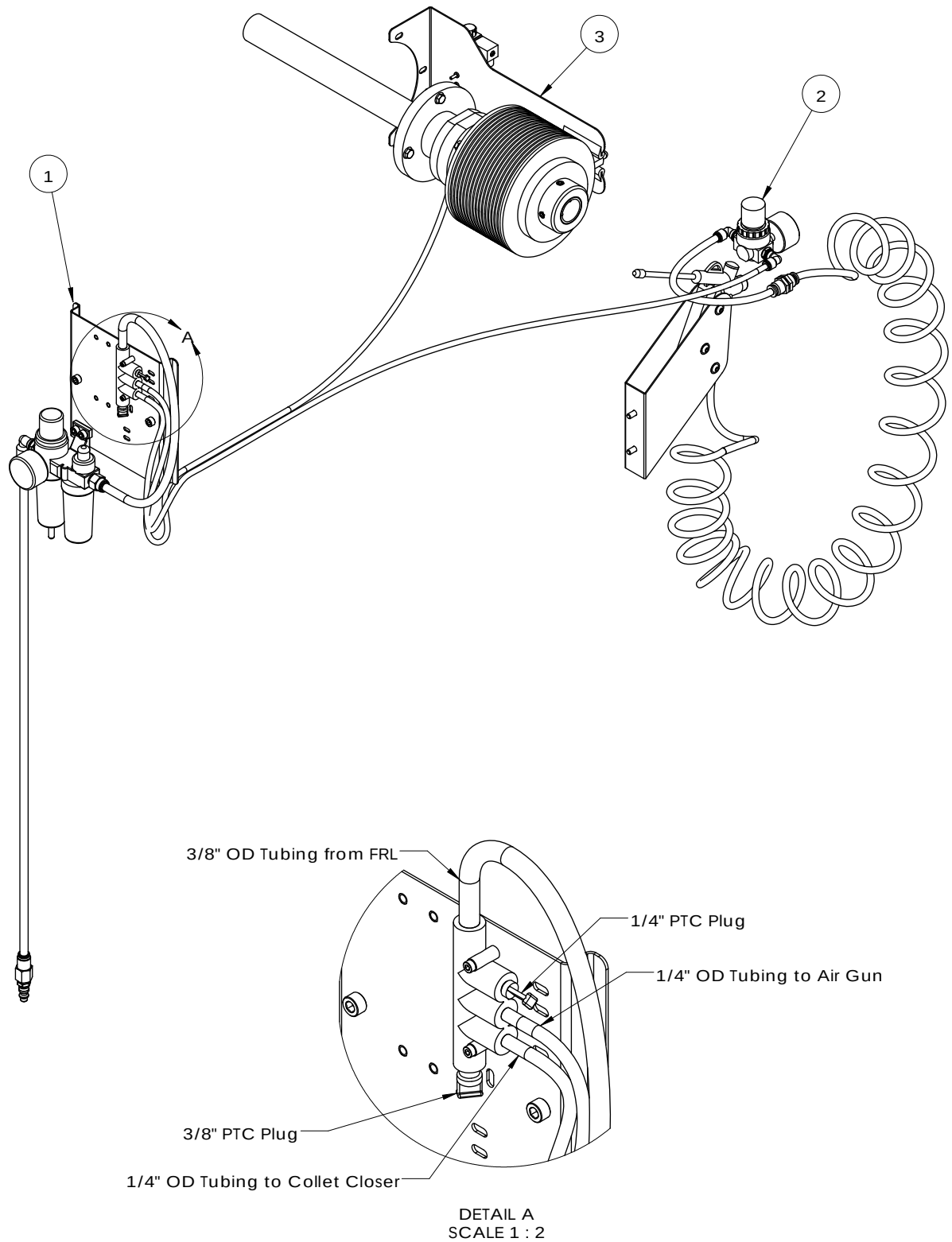
13: DIAGRAMS AND PARTS LISTS

13.12 Accessory Diagrams and Parts Lists

13.12 ACCESSORY DIAGRAMS AND PARTS LISTS

13.12.1 Pneumatics Diagram.....	249
13.12.2 Automatic Collet Closer Assembly.....	251
13.12.3 Air Gun Kit Assembly.....	253

13.12.1 Pneumatics Diagram

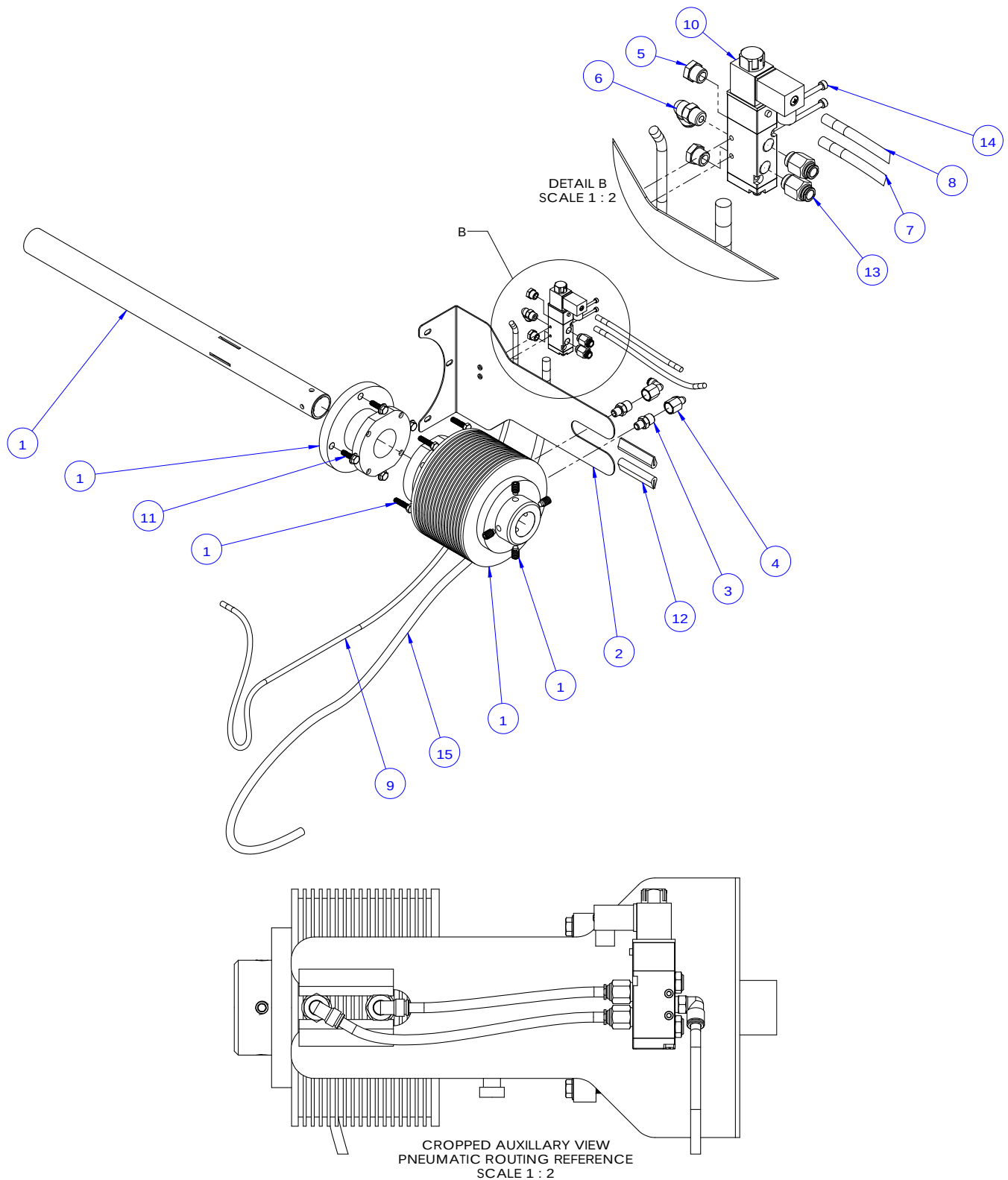


13: DIAGRAMS AND PARTS LISTS

13.12 Accessory Diagrams and Parts Lists

ID	Description	Quantity
1	Machine Pneumatics Distribution Assembly, 15LX (PN 51626)	1
2	Air Gun Kit (PN 51443)	1
3	Automatic Collet Closer for 15LX (PN 54899)	1

13.12.2 Automatic Collet Closer Assembly

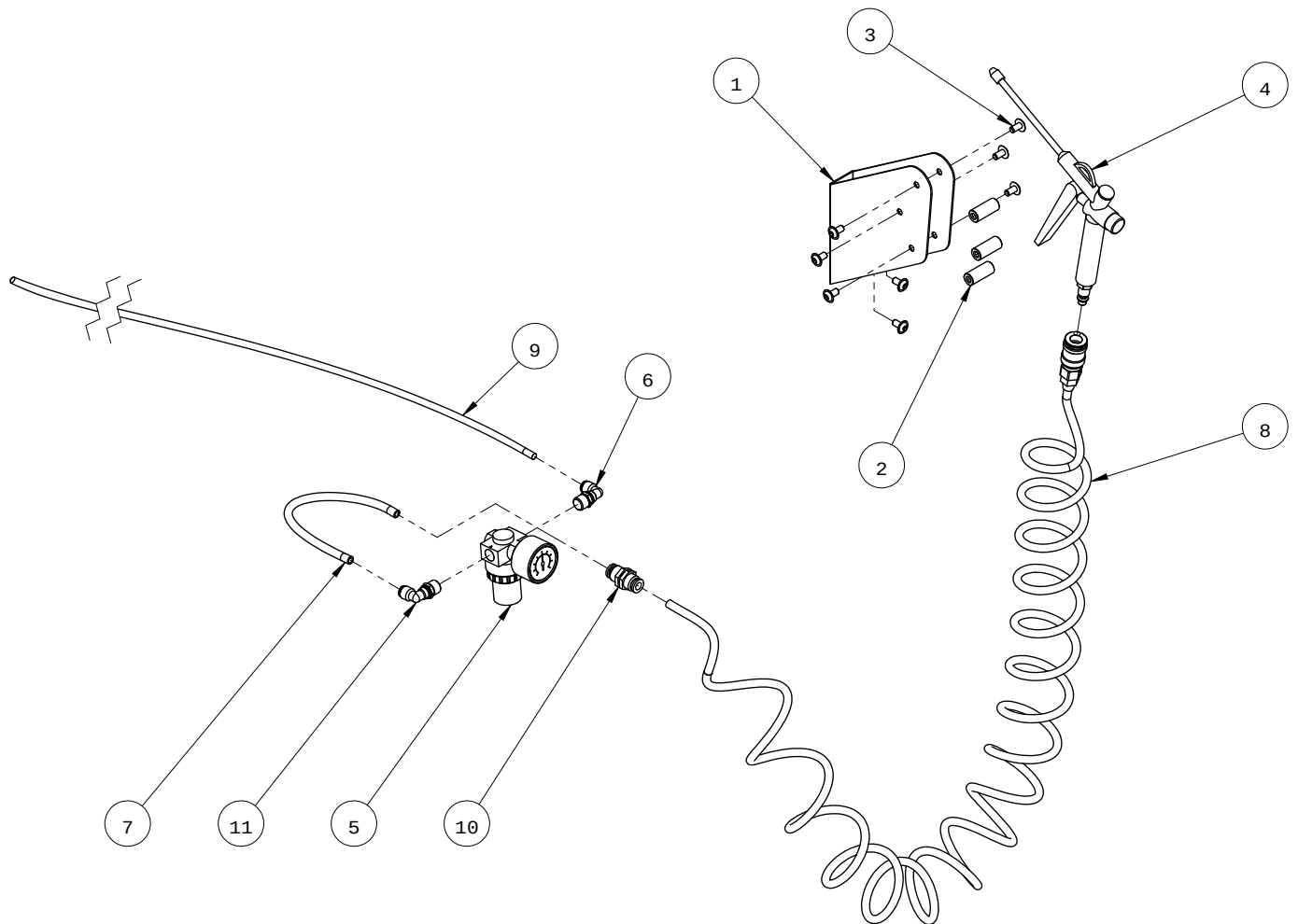


13: DIAGRAMS AND PARTS LISTS

13.12 Accessory Diagrams and Parts Lists

ID	Description	Alternative Part Number	Quantity
1	Pneumatic Collet Closer Assembly (PN 38252)	—	1
2	Anti-Rotation Bracket, 15LX Automatic Collet Closer (PN 54686)	—	1
3	Fitting, 1/4 × 1/8 NPT Male Reducer (PN 38689)	—	2
4	Fitting, Elbow, 1/4 in. Tube - 1/4 in. NPT (Female) (PN 31990)	McMaster-Carr: 5779K635	2
5	Fitting, Muffler (Flat), 1/8 NPT (PN 37297)	—	2
6	Fitting, Elbow, 1/8 in. NPT (Male) - 1/4 in. PTC (PN 34886)	—	1
7	Tubing, Polyethylene Air Line, 1/4 in. OD, Black (PN 31457)	McMaster-Carr: 5180K71	20 cm
8	Tubing, Polyethylene Air Line, 1/4 in. OD, Black (PN 31457)	McMaster-Carr: 5180K71	20 cm
9	Tubing, Polyethylene Air Line, 1/4 in. OD, Black (PN 31457)	McMaster-Carr: 5180K71	2.7 m
10	Five Port, Four Way, Two Position, 24 VDC, Single Solenoid Valve, 1/8 in. (Inlet/Outlet), 1/8 in. (Exhaust) (PN 52306)	—	1
11	Screw, Hex Head Cap (Flanged), M6 × 1 - 20, Black Oxide (PN 55254)	McMaster Carr: 95735A148	4
12	Rubber Edge Gasket (PN 54573)	—	2
13	Pneumatic, Straight, 1-8 NPT, Male × 1-4 Push-to-Connect Fitting (PN 53043)	—	2
14	Screw, Socket Head Cap, M3 × 0.5 - 30 (PN 38732)	McMaster Carr: 91290A130	2

13.12.3 Air Gun Kit Assembly



13: DIAGRAMS AND PARTS LISTS

13.12 Accessory Diagrams and Parts Lists

ID	Description	Alternative Part Number	Quantity
1	Air Gun Holster (PN 51368)	—	1
2	Air Gun Holster Spacer (PN 51442)	—	3
3	Screw, Button Head Cap (Flanged), M6 × 1 - 12, Stainless Steel (PN 38206)	McMaster-Carr: 97654A380	8
4	Air Blow Gun (PN 51435)	—	1
5	Pressure Regulator, 1/4 n. NPT × 1/4 in. NPT, 1/8 in. NPT Gauge Port, 7-125 psi, 150 psi (PN 39356)	—	1
6	<u>Fitting, Elbow, 1/4 NPT to 1/4 in. PTC (Male) (PN 31324)</u>	—	1
7	Tubing, Polyethylene Air Line, 8 mm OD, Black (PN 50707)	McMaster-Carr: 5180K71	28 cm
8	Hose Assembly (Retracting), 8 mm OD × 4 ft, Industrial Quick Connect Fitting (one end), Polyurethane, Black (PN 51524)	—	1
9	Tubing, Polyethylene Air Line, 1/4 in. OD, Black (PN 31457)	McMaster-Carr: 5180K71	250 cm
10	Fitting, Bulkhead Union, 8 mm PTC - 8 mm PTC (PN 52441)	—	1
11	Fitting, Elbow, 1/4 in. NPT (Male) - 8 mm PTC (PN 52444)	—	1

Notes:

PN 51435 Includes a male industrial quick connect fitting.

PN 51524 includes a female industrial quick connect fitting on one end of the tube.

ELECTRICAL SCHEMATICS

IN THIS SECTION, YOU'LL LEARN:

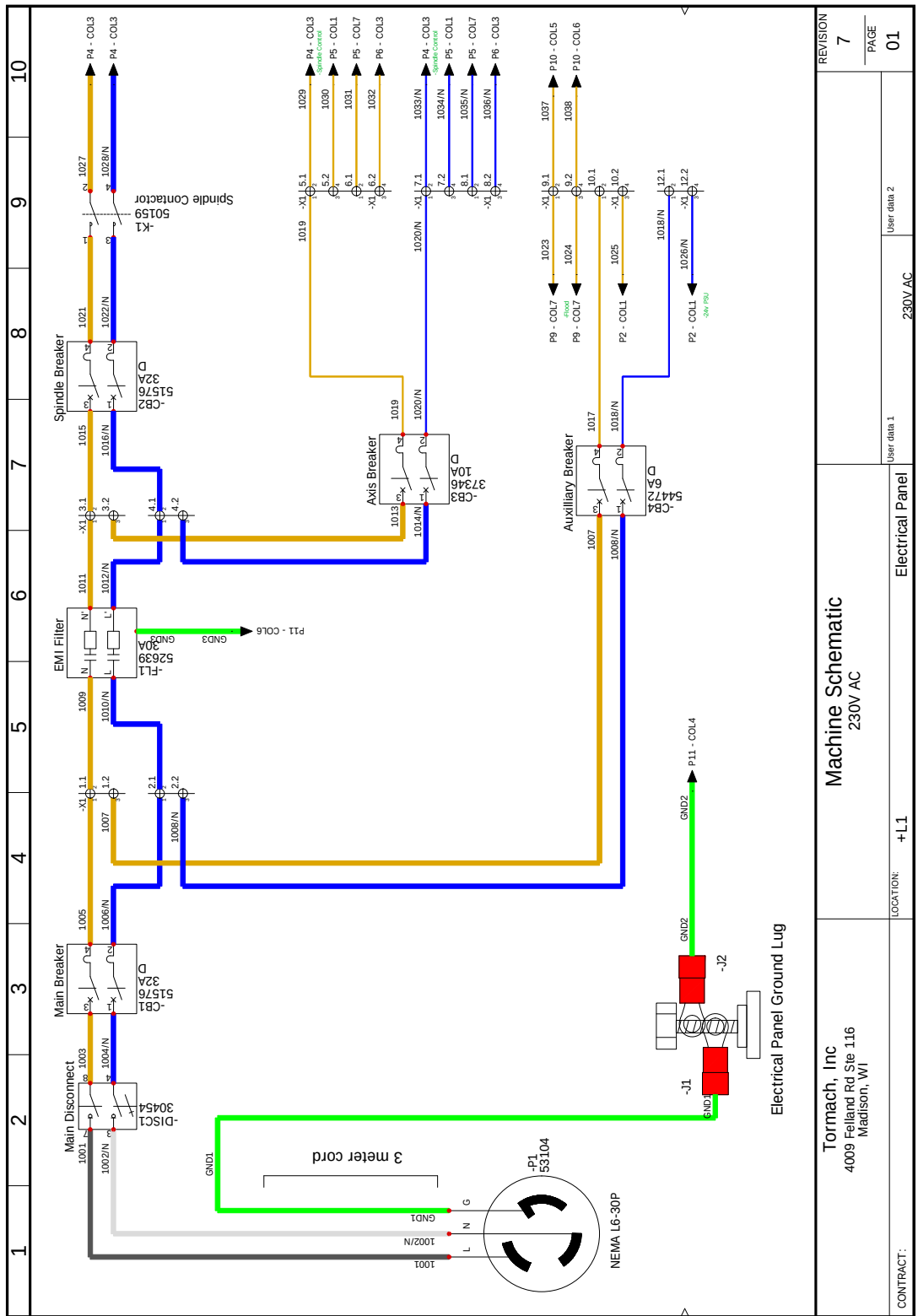
- About the electrical schematics for this machine's electronics.

CONTENTS

14.1 230 Vac Power (Sheet 1)	256
14.2 24 Vdc Power (Sheet 2)	257
14.3 Safety Systems (Sheet 3)	258
14.4 Spindle Drive (Sheet 4)	259
14.5 XZ Axis Drives (Sheet 5)	260
14.6 Turret Drive (Sheet 6)	261
14.7 IO Board – Power, Safety, Aux Outputs (Sheet 7)	262
14.8 IO Board – Digital Inputs and Outputs (Sheet 8)	263
14.9 IO Board – Aux Inputs, Brakes, Coolant (Sheet 9)	264
14.10 Accessory / Aux (Sheet 10)	265
14.11 Grounds (Sheet 11)	266
14.12 AC Terminal Strip (-X1) (Sheets 12-14)	267
14.13 Control Terminal Strip (-X2) (Sheet 15)	270
14.14 Grounds Terminal Strip (-X3) (Sheet 16)	271
14.15 Bill of Materials (Sheet 17-18)	272

14.1 230 Vac Power (Sheet 1)

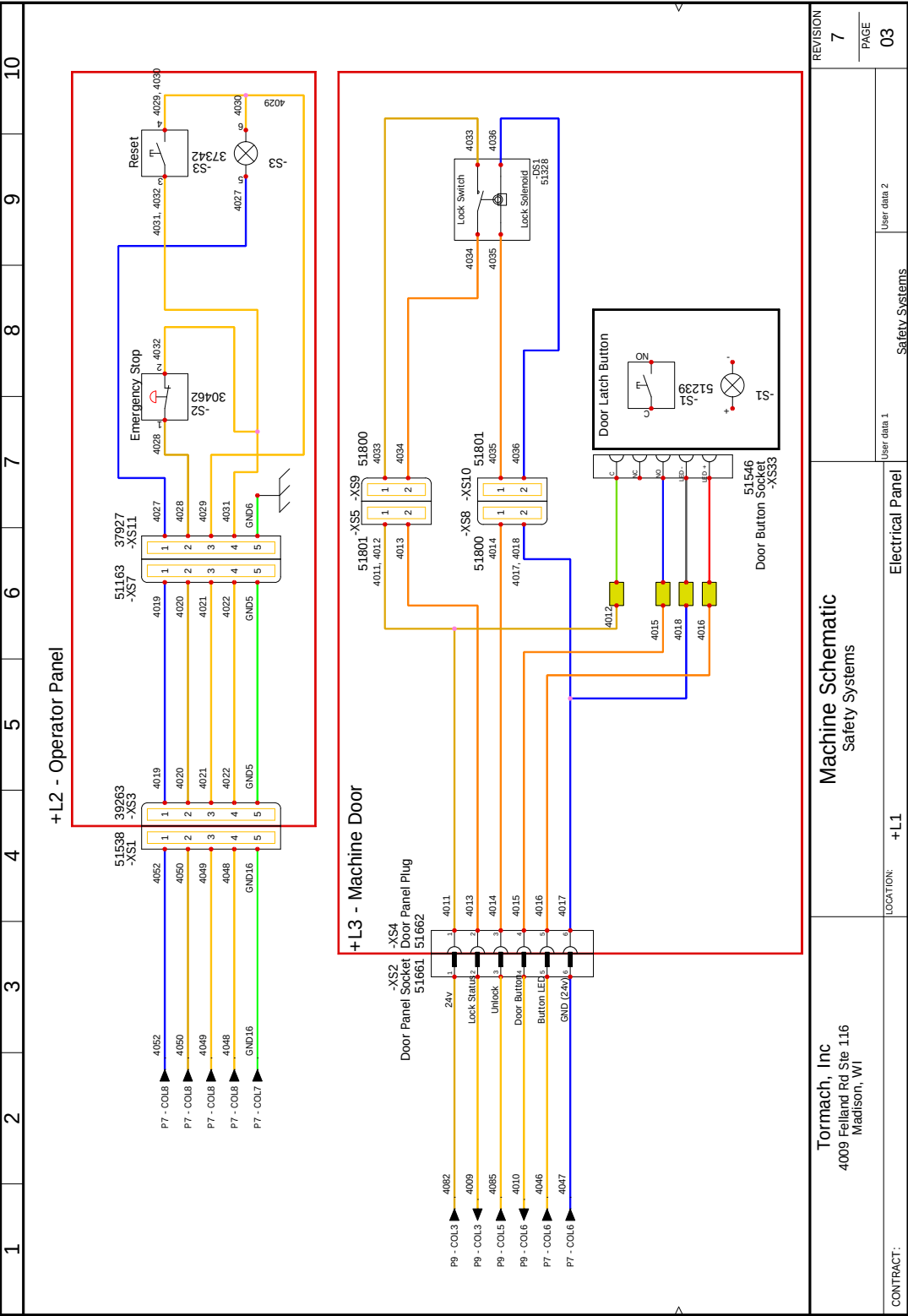
14.1 230 VAC POWER (SHEET 1)



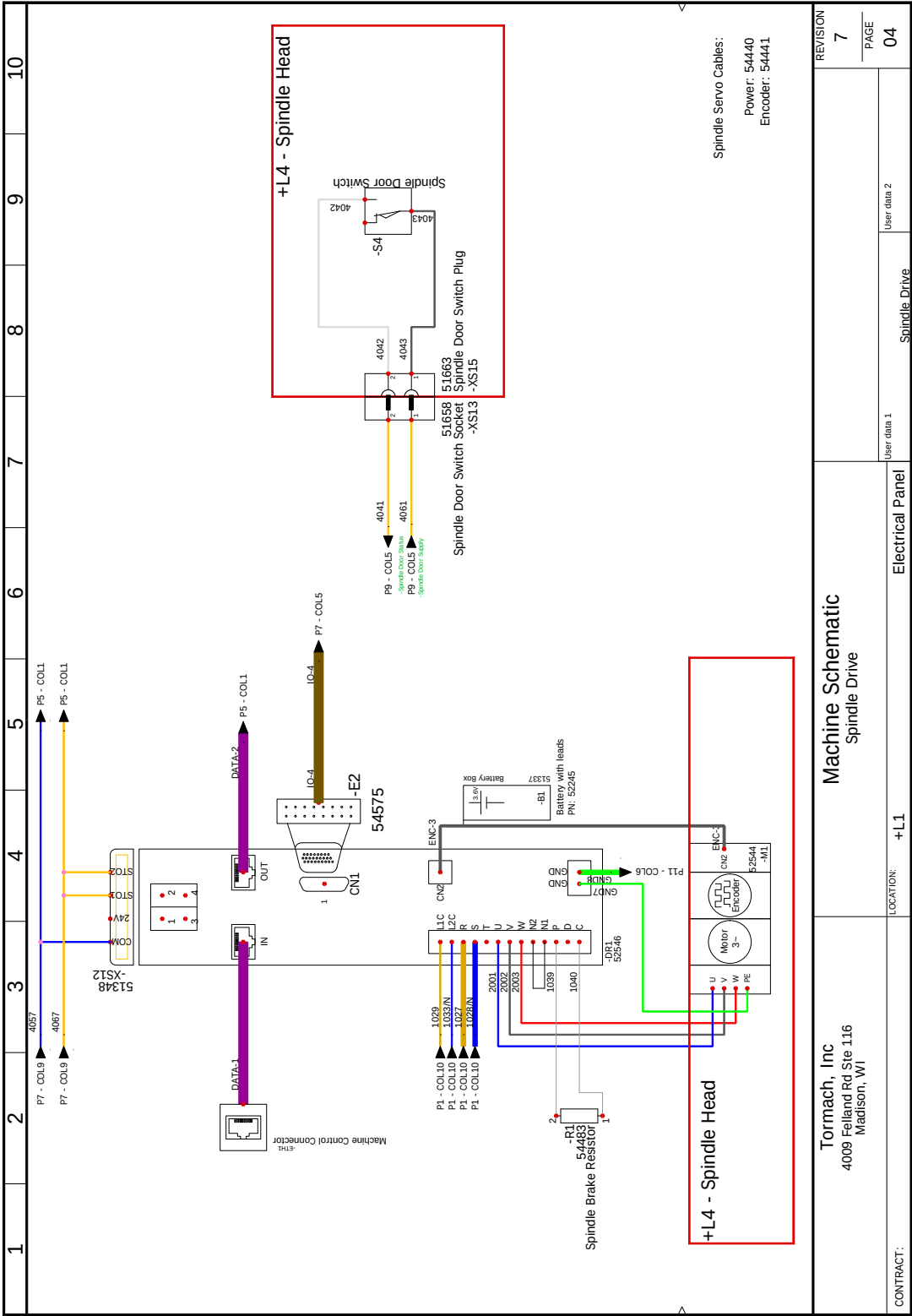
1	2	3	4	5	6	7	8	9	10																		
The diagram illustrates the 24V DC power system. It begins with a 24V DC Power Supply (PS1, 54485) connected to an 85-264 VAC source. The supply's output is split into three main paths: a yellow path (4051, 4001, 4005) leading to a P7 - COL2 terminal; a blue path (4002, 4008, 4007) leading to P7 - COL2 and P7 - COL1 terminals; and a red path (4003, 4004) leading to the fan and the spindle motor. The Electrical Cabinet Fan (37526) is connected to the red path. A Thermal Switch (-TS1, 3742Q) is connected to the yellow path and the spindle motor. The Spindle Contactor (K1, 50159) is connected to the red path and the spindle motor. The diagram also includes a legend for wire colors and a table of wire numbers. <table border="1"> <thead> <tr> <th>Wire Number</th> <th>Color</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>Yellow</td> </tr> <tr> <td>2</td> <td>Blue</td> </tr> <tr> <td>3</td> <td>Red</td> </tr> <tr> <td>4</td> <td>Blue</td> </tr> <tr> <td>5</td> <td>Green</td> </tr> <tr> <td>6</td> <td>Blue</td> </tr> <tr> <td>13</td> <td>Blue</td> </tr> <tr> <td>14</td> <td>Blue</td> </tr> </tbody> </table>										Wire Number	Color	1	Yellow	2	Blue	3	Red	4	Blue	5	Green	6	Blue	13	Blue	14	Blue
Wire Number	Color																										
1	Yellow																										
2	Blue																										
3	Red																										
4	Blue																										
5	Green																										
6	Blue																										
13	Blue																										
14	Blue																										
Tormach, Inc 4009 Felland Rd Ste 116 Madison, WI			Machine Schematic 24V DC Power				Electrical Panel																				
CONTRACT:			LOCATION: +L1			User data 1			User data 2																		
REVISION			7			PAGE			02																		

14: ELECTRICAL SCHEMATICS
14.3 Safety Systems (Sheet 3)

14.3 SAFETY SYSTEMS (SHEET 3)



14.4 SPINDLE DRIVE (SHEET 4)



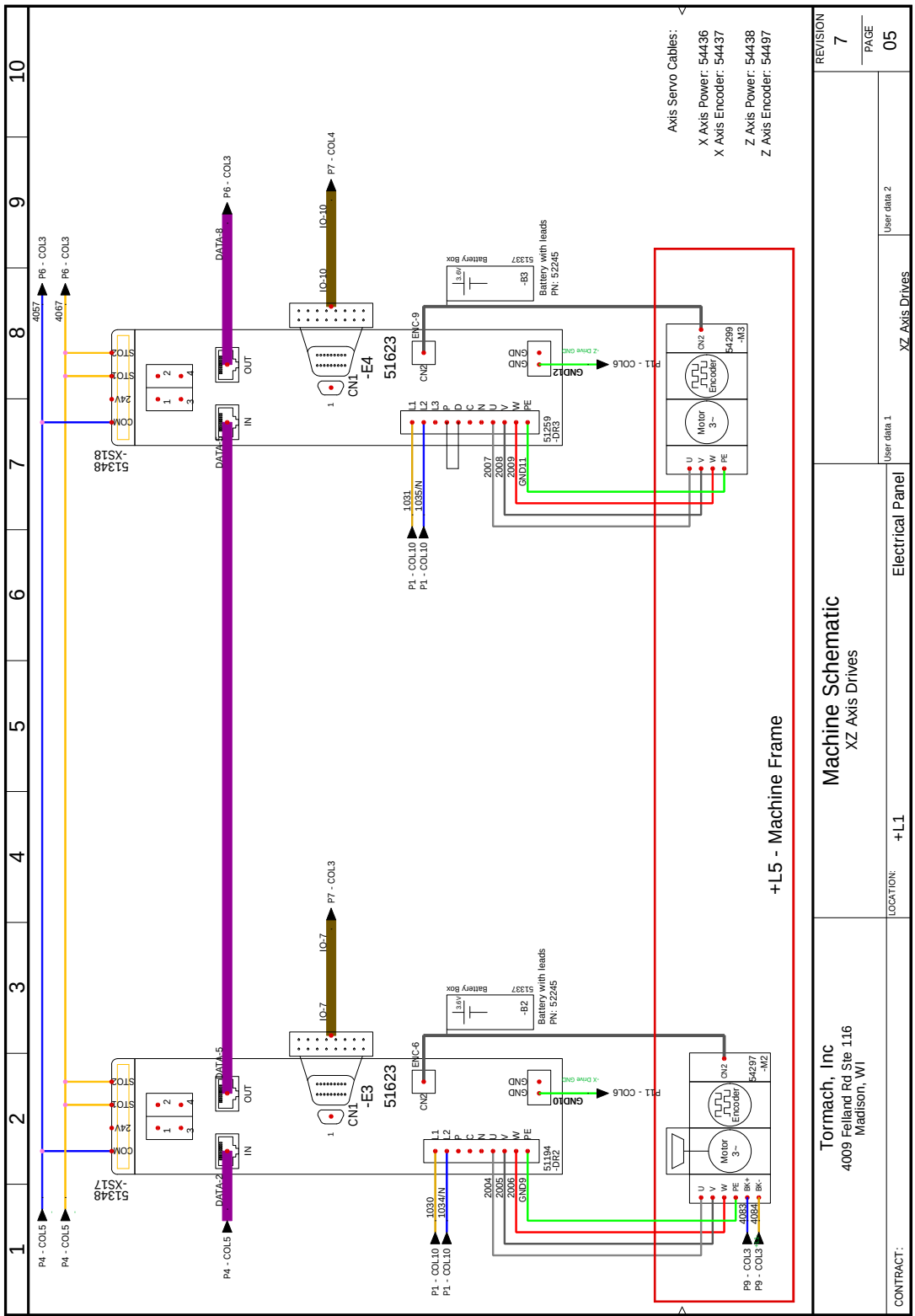
SOLIDWORKS Electrical

Tormach, Inc 4009 Folland Rd Ste 116 Madison, WI	Machine Schematic Spindle Drive		REVISION	
	LOCATION: +L1	Spindle Drive	7	04
CONTRACT:	User data 1		User data 2	

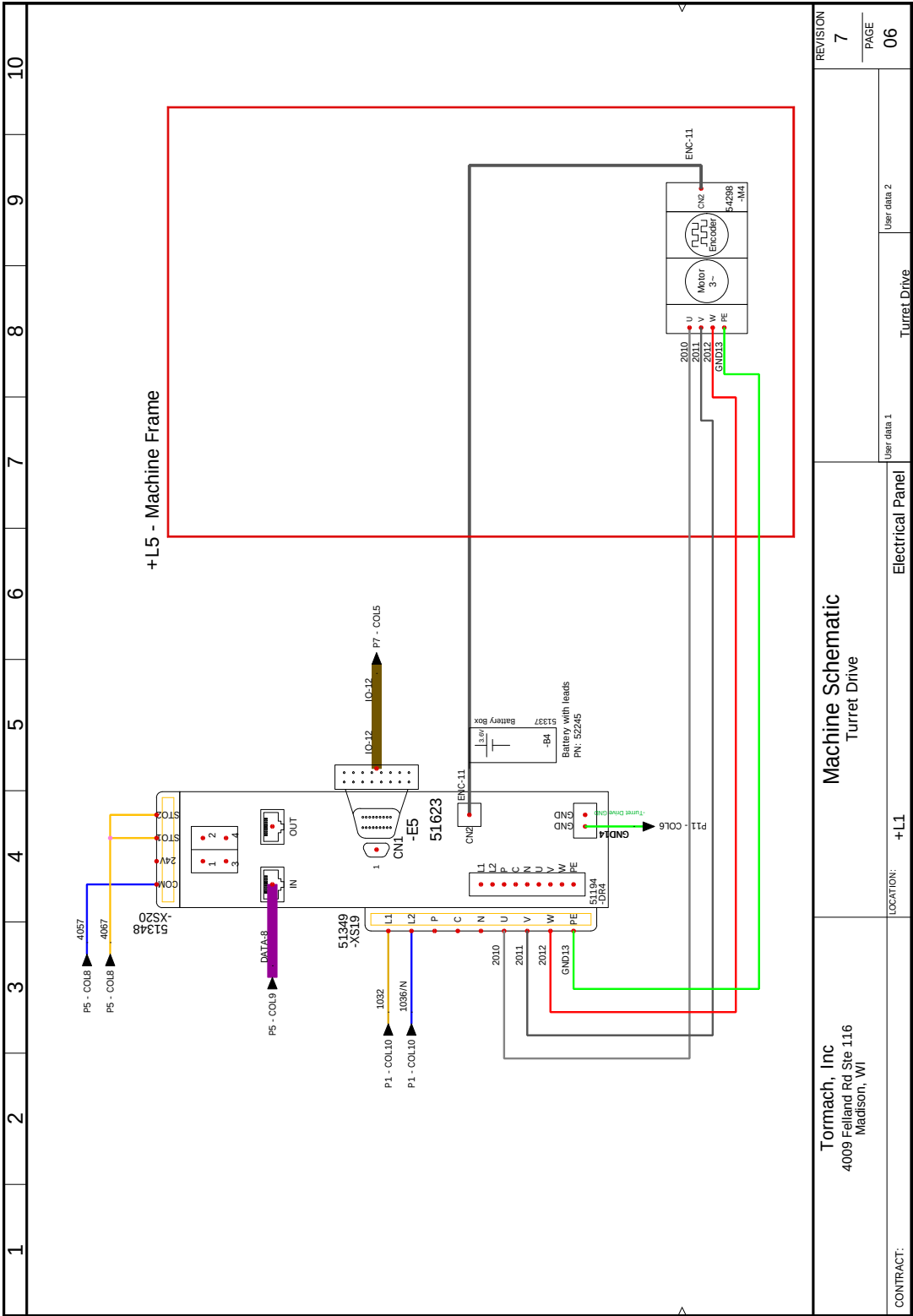
Document created with version: 2022.5.0.00063

14.5 XZ Axis Drives (Sheet 5)

14.5 XZ AXIS DRIVES (SHEET 5)

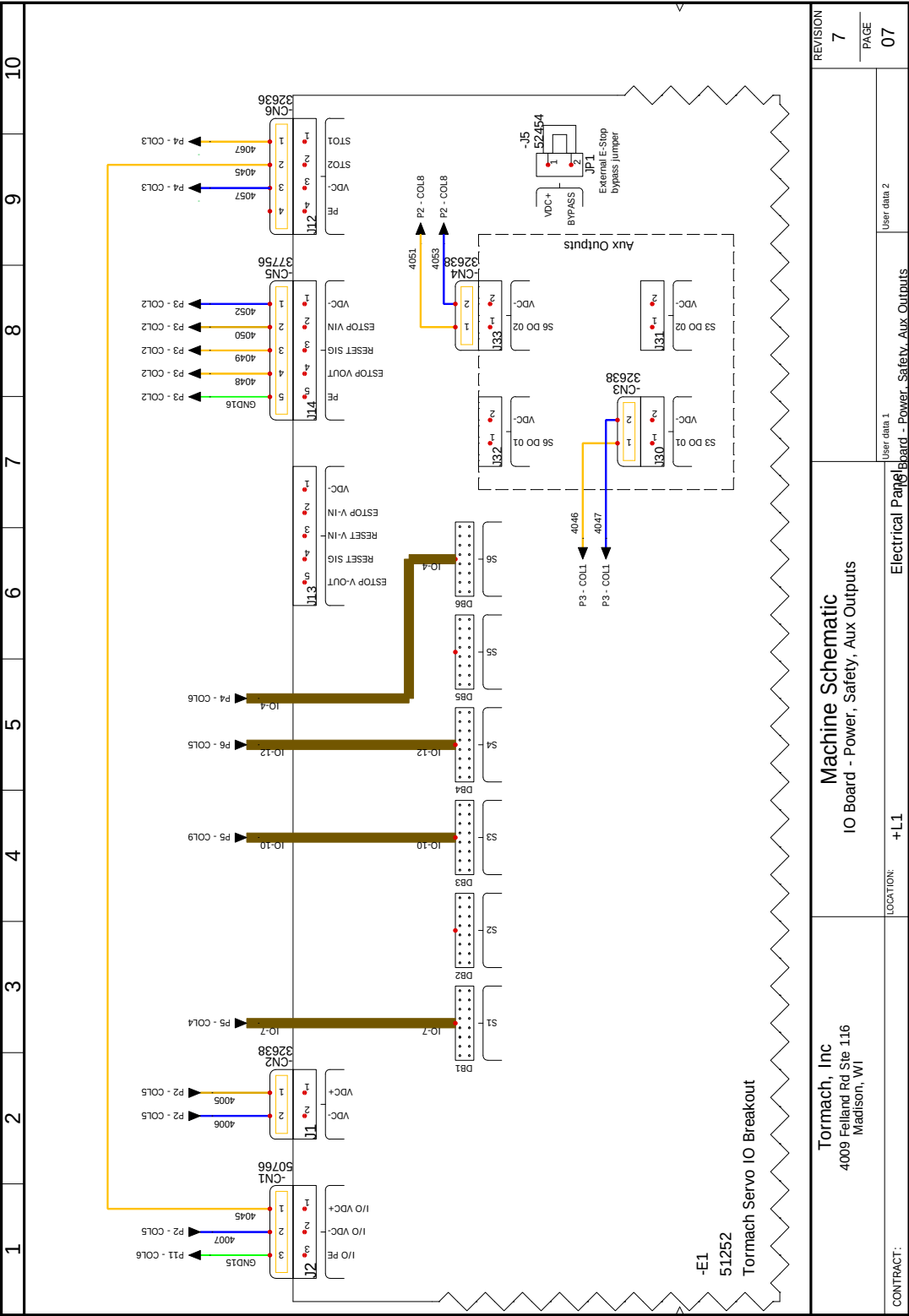


14.6 TURRET DRIVE (SHEET 6)



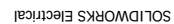
14: ELECTRICAL SCHEMATICS
14.7 IO Board – Power, Safety, Aux Outputs (Sheet 7)

14.7 IO BOARD – POWER, SAFETY, AUX OUTPUTS (SHEET 7)



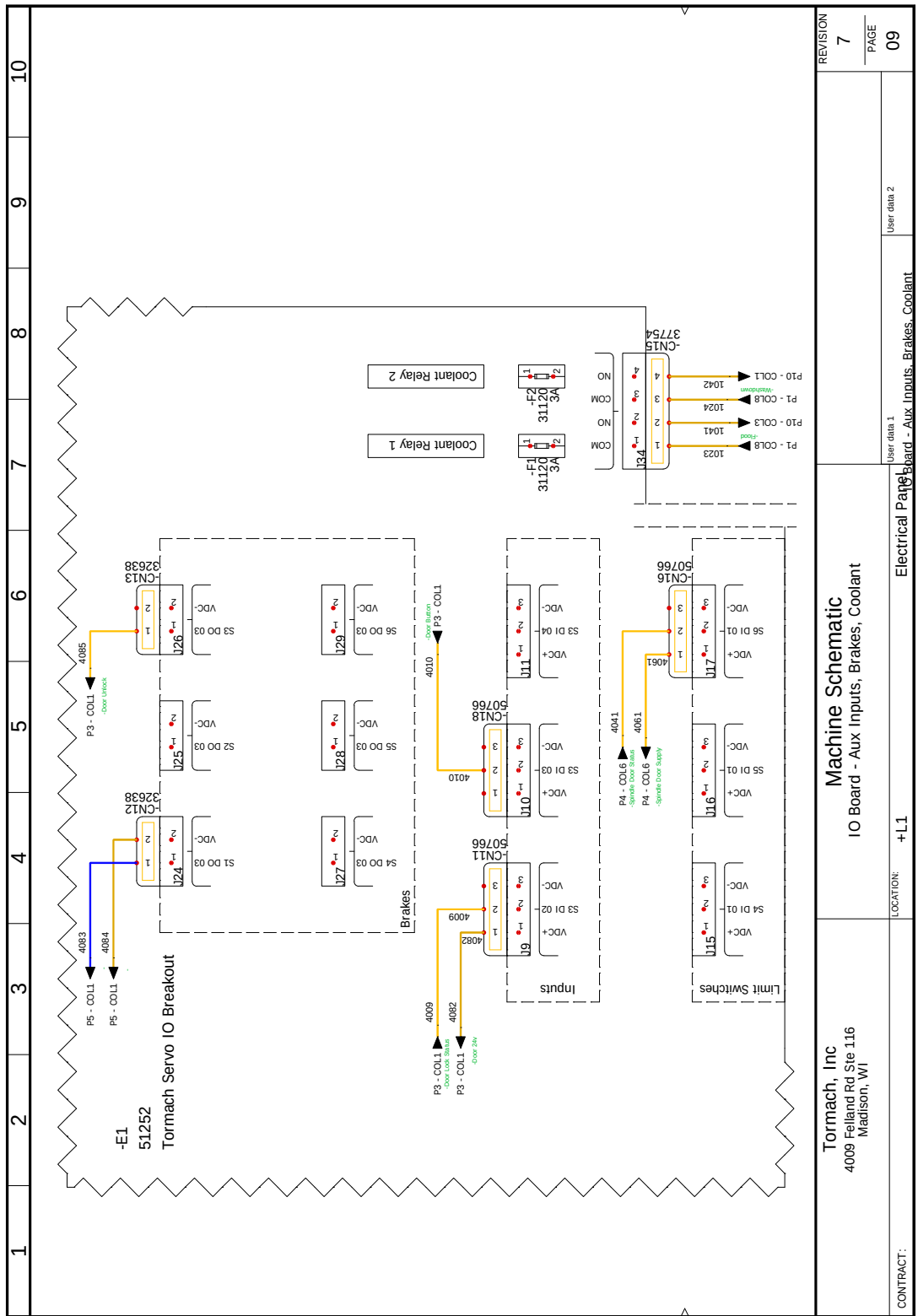
14.8 IO Board – Digital Inputs and Outputs (Sheet 8)

14.8 IO BOARD – DIGITAL INPUTS AND OUTPUTS (SHEET 8)

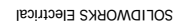


14.9 IO Board – Aux Inputs, Brakes, Coolant (Sheet 9)

14.9 IO BOARD – AUX INPUTS, BRAKES, COOLANT (SHEET 9)



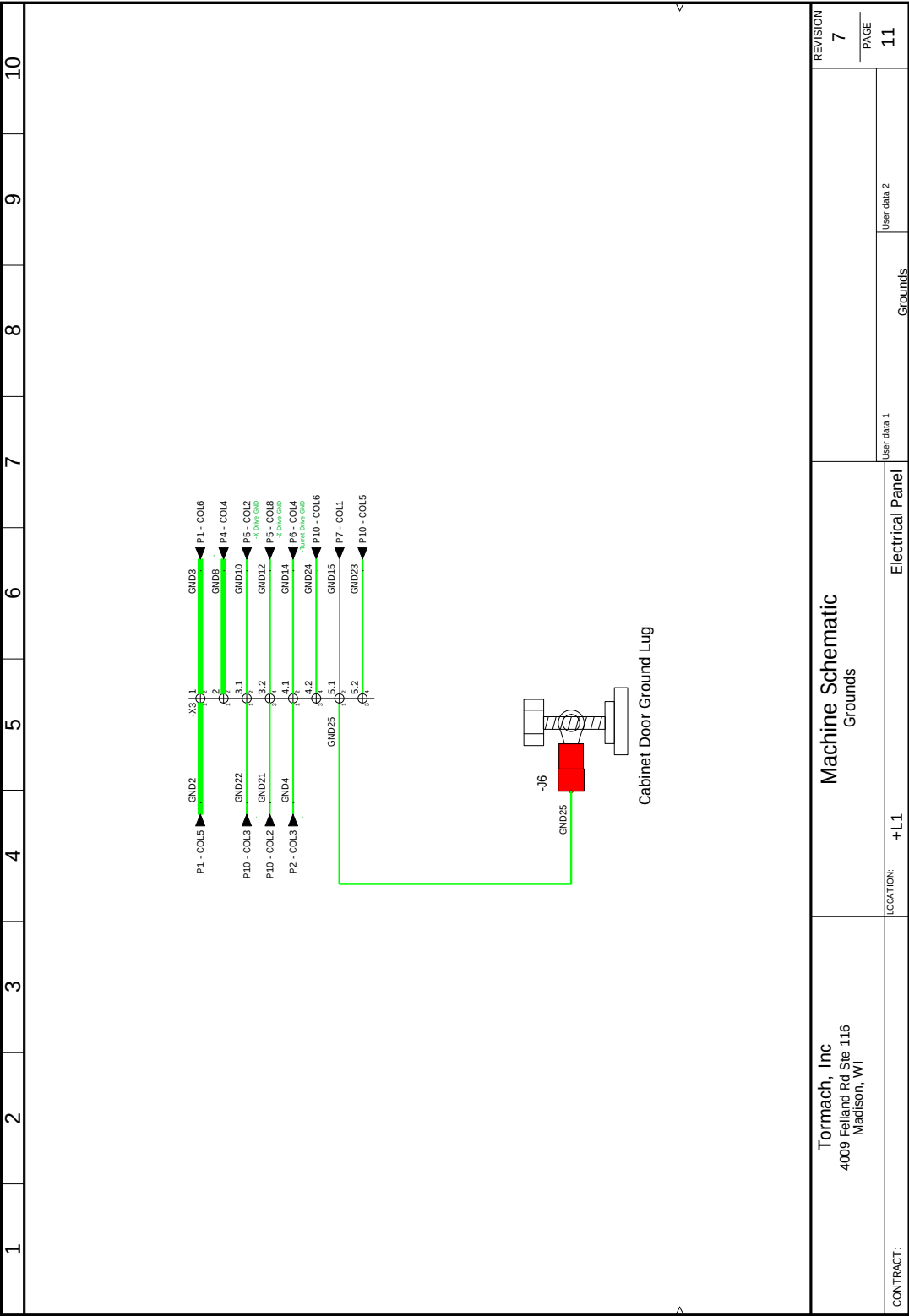
14.10 ACCESSORY / AUX (SHEET 10)



14: ELECTRICAL SCHEMATICS

14.11 Grounds (Sheet 11)

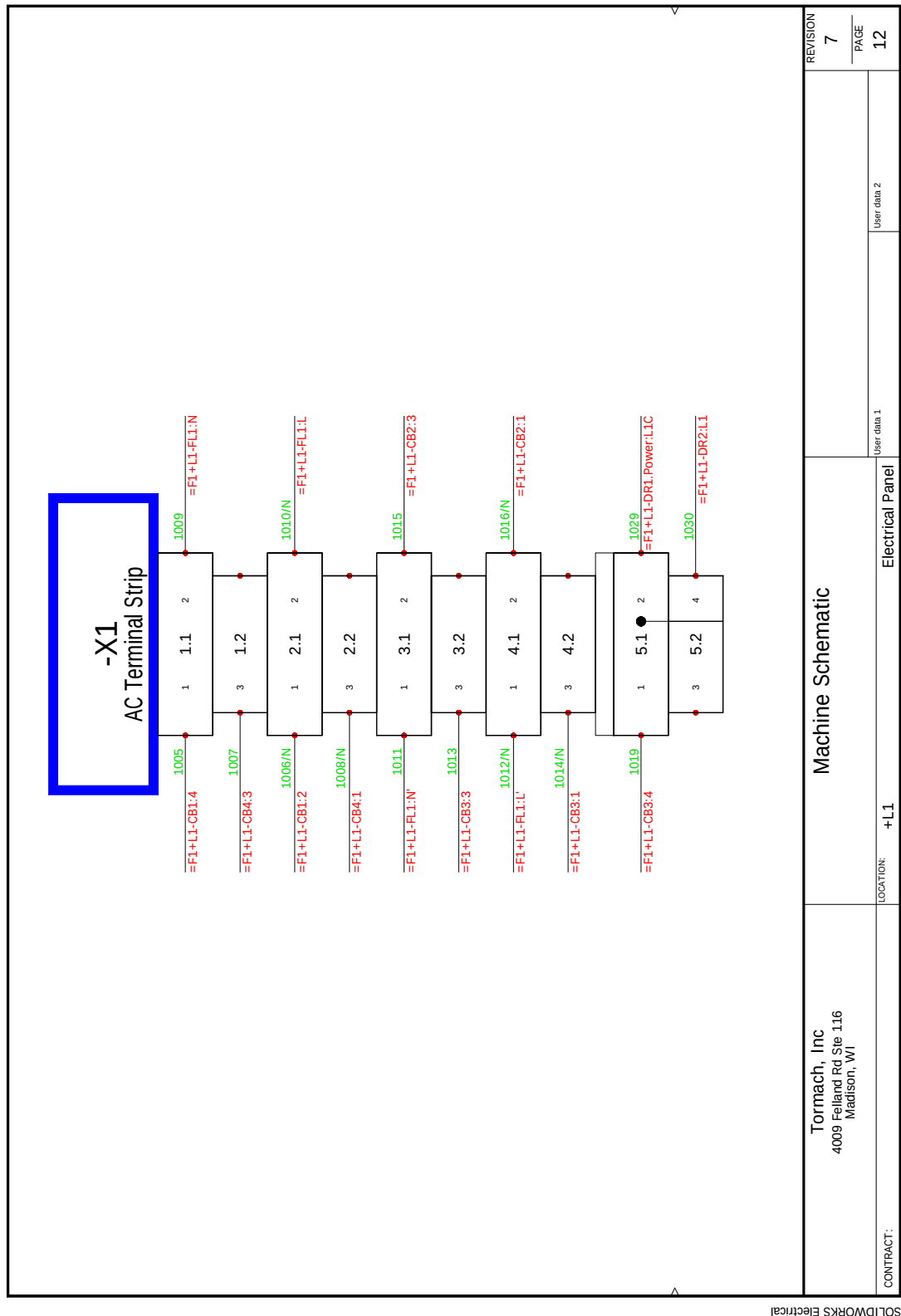
14.11 GROUNDS (SHEET 11)



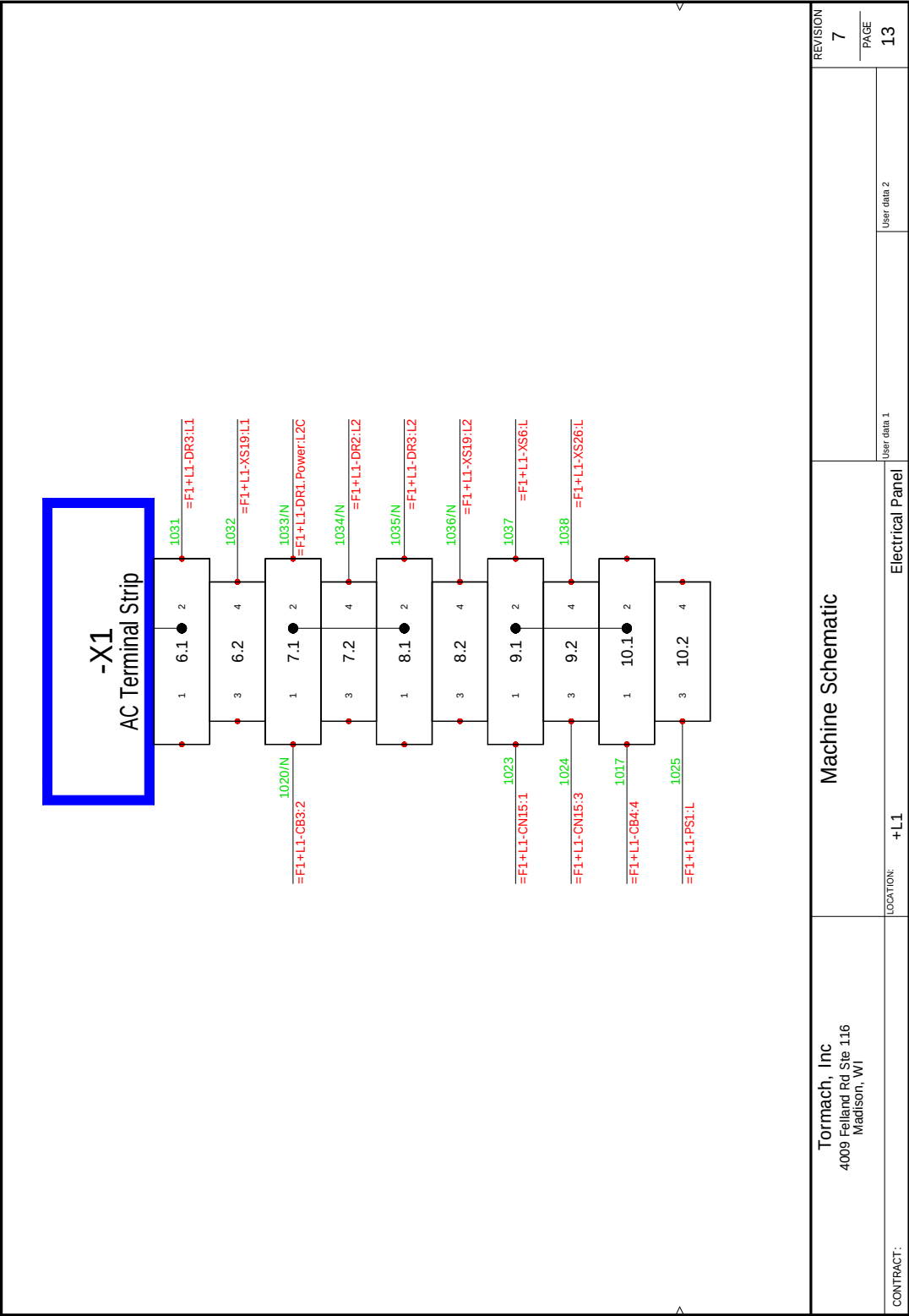
SOLIDWORKS Electrical

Document released with version: 2025.1.1.0.00153

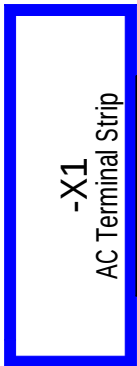
14.12 AC TERMINAL STRIP (-X1) (SHEETS 12-14)



14: ELECTRICAL SCHEMATICS
14.12 AC Terminal Strip (-X1) (Sheets 12-14)



14.12 AC Terminal Strip (-X1) (Sheets 12-14)

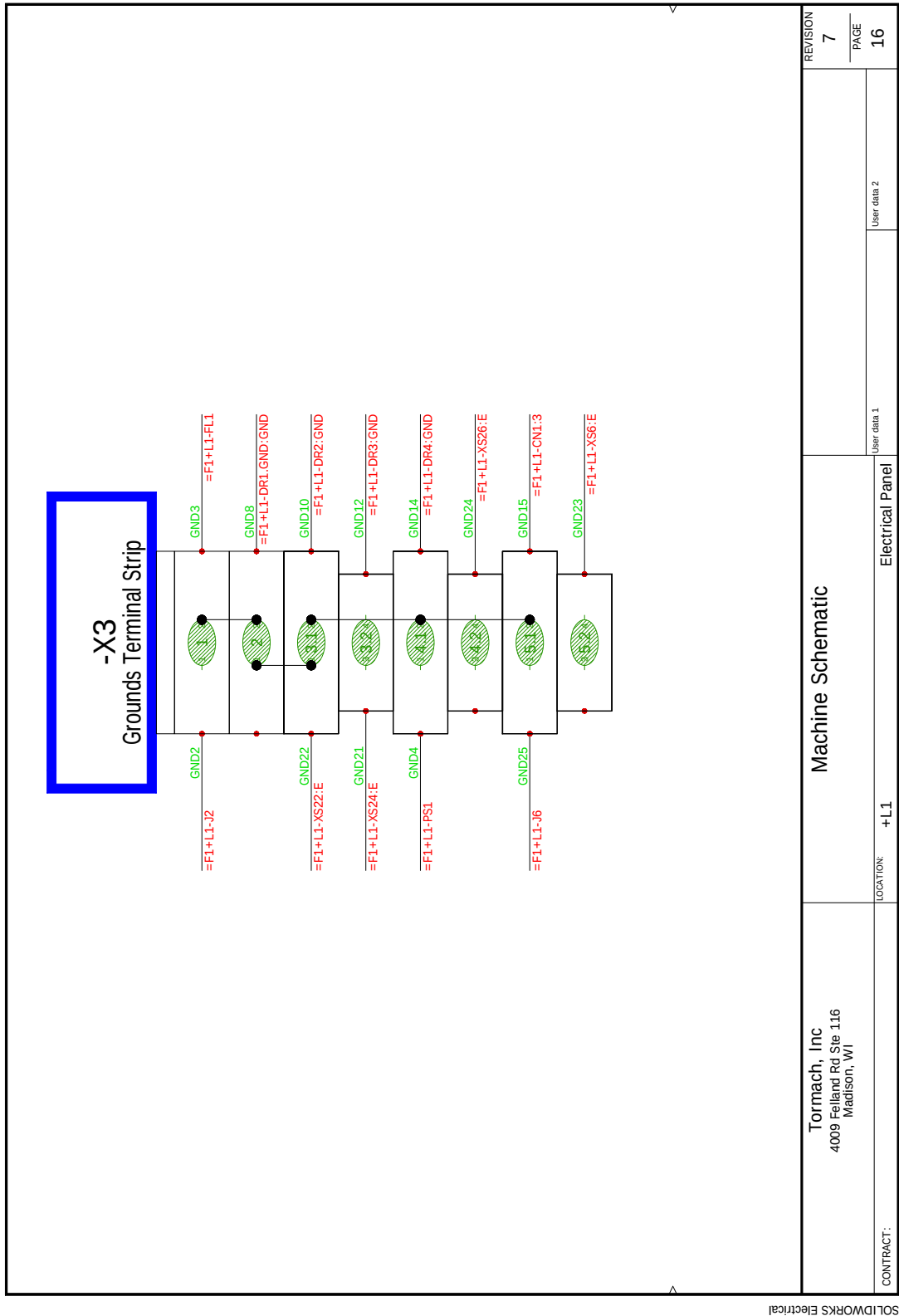


Tormach, Inc
4009 Felland Rd Ste 116
Madison, WI

Document realized with version : 2025.5.0.0053

14.13 Control Terminal Strip (-X2) (Sheet 15)

14.14 GROUNDS TERMINAL STRIP (-X3) (SHEET 16)



14: ELECTRICAL SCHEMATICS

14.15 Bill of Materials (Sheet 17-18)

14.15 BILL OF MATERIALS (SHEET 17-18)

Reference	Mark	Location	Location description	Description	Quantity
52454	-J5	+L1	Electrical Panel	Jumper, Two Position, Female, 0.100IN Pitch	1
50159	-X1	+L1	Electrical Panel	Contactor Relay, 3 position 220V 3P, 24V DC Coil, Screw Terminal	1
32636	-C06	+L1	Electrical Panel	Terminal block (plug) 4 position horizontal, Dagon	1
32638	-CN2, -CN3, -CN4, -CN12, -CN13	+L1	Electrical Panel	Terminal block (plug) 2 position, Dagon	5
37764	-CN15	+L1	Electrical Panel	Term Block Plug, 4pos, 5.08mm	1
37756	-C05	+L1	Electrical Panel	Terminal block (plug) 5 position, Dagon	1
50766	-CN1, -CN11, -CN16, -CN18	+L1	Electrical Panel	Terminal block (plug) 3 position, Dagon	4
31120	-F1, -F2	+L1	Electrical Panel	Fuse, 3A, 250V, 5x20, Glass, Slow-Blow	2
51328	-DS1	+L3	Machine Door	Locking Door Switch	1
30454	-D15C1	+L1	Electrical Panel	Main Disconnect Switch	1
37346	-CB3	+L1	Electrical Panel	Circuit Breaker, 2-Pole, 10A, 230/400VAC, D Curve, DIN Mount	1
51576	-CB1, -CB2	+L1	Electrical Panel	Circuit Breaker, 2-Pole, 32A, 230/400VAC, D Curve, DIN Mount	2
54472	-CB4	+L1	Electrical Panel	Circuit Breaker, 2-Pole, 6A, 230/400VAC, D Curve, DIN Mount	1
53104	-P1	+L1	Electrical Panel	Plug, Male, 250 VAC, 30 Amps, Locking Connector, NEMA L6-30P	1
51184	-DR2	+L1	Electrical Panel	SV650N EtherCat AC Servo Drive, Size A	1
51259	-DR3	+L1	Electrical Panel	SV650N EtherCat AC Servo Drive, Size B	1
51337	-B1, -B2, -B3	+L1	Electrical Panel	Inovance SV650N Battery Box	3
51348	-XS20	+L1	Electrical Panel	Inovance STO Connector	1
51349	-XS19	+L1	Electrical Panel	CON SV650A 9P	1
52544	-M1	+L4	Spindle Head	Servo Motor, 3kW, 3000RPM, 130mmx130mm Frame, 9.8 N.m, CE & UL Listed	1
52546	-M2	+L1	Electrical Panel	SV670N EtherCat AC Servo Drive, Size E	1
54287	-M3	+L5	Machine Frame	Servo Motor, 400W, 220V, 2.4A, 3K RPM (7K Max), Oil Seal and Brake, R Sub-series	1
54299	-M5	+L5	Machine Frame	Servo Motor, 750W, 220V, 4.4A, 3K RPM (7K Max), Oil Seal, R Sub-series	1
52639	-F11	+L1	Electrical Panel	EMI Filter	1
53800	-XS8	+L3	Machine Door	Connector, Plug, 2-pin Female JST, 2.5mm Pitch	1
53801	-XS5	+L3	Machine Door	Connector, Receptacle, 2-pin Male JST, 2.5mm Pitch	1
51657	-CN7, -CN8, -CN9, -CN10	+L1	Electrical Panel	Rectangular Connector, 5 Pin Female, 0.100" (2.54mm) Pitch	4
51658	-XS13	+L1	Electrical Panel	Rectangular Connector, 2 Pin Male Vertical, Plug, White 0.185" (4.2mm)	1
51661	-XS2	+L1	Electrical Panel	Rectangular Connector, 6 Pin Male Dual Row, Plug, White 0.185" (4.2mm)	1
51662	-XS4	+L3	Machine Door	Rectangular Connector, 6 Pin Female Dual Row, Receptacle, White 0.185" (4.2mm)	1
51663	-XS15	+L4	Spindle Head	Rectangular Connector, 2 Pin Female Vertical, Receptacle, White 0.185" (4.2mm)	1
54485	-PS1	+L1	Electrical Panel	DIN Rail Power Supply, 85-264 VAC, 24V, 120W, 5A, 40mm*123.6mm	1
52245	-B1, -B2, -B3	+L1	Electrical Panel	SV650N Encoder Battery Assembly	3

Tormach, Inc
4009 Felland Rd Ste 116
Madison, WI

Machine Schematic

+L1

Electrical Panel

REVISION
7

PAGE
17

Document released with version: 2024.6, E.O. 14176

Reference	Mark	Location	Location description	Description	Quantity
51559	-XS26	+L1	Electrical Panel	Power Outlet (IEC 320, C13 Double Socket), Spade Terminals	1
51560	-XS6 , -XS2 , -XS24	+L1	Electrical Panel	Power Outlet (IEC 320-2-2/F, C13 Single Socket), Spade Terminals	3
37365	-ETH1	+L1	Electrical Panel	RJ45 Bulkhead Connector (Ethernet)	1
37420	-TS1	+L1	Electrical Panel	Thermal Switch (Pushbutton Reset)	1
54483	-R1	+L1	Electrical Panel	Brake Resistor, 20 Ohm, 300 Watt, 20' leads	1
51239	-S1	+L3	Machine Door	Pushbutton, JNO3NC, White LED	1
51546	-XS33	+L3	Machine Door	Button Wiring Pigtail, 5-pin, For Tuyen TMB22 Buttons	1
51293	-XS21 , -XS23	+L1	Electrical Panel	M12 Socket, Panel feed through, for rear mounting, A-coding, Crimp termination, 5 pin	2
51543	-XS26 , -XS27	+L1	Electrical Panel	M12 Plug, Panel Mount, Female Pins, 4 Position, A Keying	2
51252	-E1		Electrical Panel	Tormach Servo IO Breakout	1
51623	-E3 , -E4 , -E5	+L1	Electrical Panel	HD15 Male Pin to IDC 2x8 Socket Adapter	3
54675	-E2	+L1	Electrical Panel	HD26 Male Pin to IDC 2x8 Socket Adapter	1
37526	-FAN1	+L1	Electrical Panel	Fan, 120mm, 24V, 0.29A	1
51289	-X3-3 , -X3-4 , -X3-5	+L1	Electrical Panel	Terminal Block, GND, 24-10AWG, 4 position, Spring Type, Internally Jumpered	3
51290	-X2-1 , -X2-2 , -X1-5 , -X1-6 , -X1-7 , -X1-8 , -X1-9 , -X1-10 , -X1-11 , -X1-12	+L1	Electrical Panel	Terminal Block, 24-10AWG, 4 position, Spring Type, Internally Jumpered	10
51291	-X1 , -X2 , -X3	+L1	Electrical Panel	Terminal Block End Plate, JUT3-4/2-2 Series	3
54474	-X1-1 , -X1-2 , -X1-3 , -X1-4	+L1	Electrical Panel	Terminal Block, 24-8AWG, 3 position, Spring Type, Internally Jumpered	4
54475	-X3-1 , -X3-2	+L1	Electrical Panel	Terminal Block, GND, 24-8AWG, 2 position, Spring Type, Internally Jumpered	2
54640	-X1	+L1	Electrical Panel	Terminal Block End Plate, JUT3-6/1-2 Series	1
51538	-XS1	+L1	Electrical Panel	Circular Connector, Front-Nut Mount Reseplace, 5 Pin Female, 20mm Cutout Round, Solder	1

Machine Schematic

+L1

Electrical Panel

REVISION
7
PAGE
18

SOLIDWORKS Electrical

Document released with version : 2002/16, 16, 0, 000/13

