

TORMACH® 1300PL®



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®, 1300PL®, 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. 2025

IMPORTANT INFORMATION: PLEASE READ FIRST

INSPECT THE ITEMS DELIVERED

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

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



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

SAVE THESE INSTRUCTIONS!

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

PURPOSE AND SCOPE OF THIS DOCUMENT

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

GETTING HELP

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

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

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

SHARE LOG DATA .ZIP FILES

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

To share log data .zip files:

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

LIABILITY DISCLAIMER

We've made every effort to provide comprehensive and accurate information, but no warranty or fitness is claimed or implied. All information provided is on an as is basis. The authors, publisher, and Tormach, Inc. ("we", "us", and so on) shall not have any liability for, or responsibility to, any person or entity for any reason for any loss or damage arising from the information contained in this document.

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

IMPORTANT INFORMATION: PLEASE READ FIRST.....	3
SAFETY.....	15
1.1 Intended Use	16
1.2 Machine Standards	17
1.2.1 American National Safety Institute (ANSI).....	17
1.2.2 Occupational Safety and Health Administration (OSHA).....	17
1.3 Safety Overview.....	18
1.4 Safety Messages.....	19
1.4.1 Personal Injury.....	19
1.4.2 Property Damage.....	19
1.5 Safety Decals.....	20
1.5.1 On the Electrical Cabinet Door.....	20
1.5.2 On the Front of the Bed	20
1.5.3 On the Gantry and the Plasma Head	21
1.6 Information Decals.....	22
1.6.1 Serial Number Plate.....	22
1.7 Machine Safety.....	23
1.7.1 General Shop Safety.....	23
1.7.2 Operational Safety.....	23
General.....	23
Workholding.....	24
1.7.3 Electrical Safety.....	24
ABOUT YOUR MACHINE.....	27
2.1 Performance Expectations.....	28
2.1.1 Cutting Performance Reference.....	28
2.1.2 Torch Height Control Reference.....	28
2.1.3 Resolution and Accuracy Reference.....	28
2.2 Machine Specifications.....	29
SITE REQUIREMENTS.....	31
3.1 General Site and Space Requirements.....	32
3.1.1 Site Requirements.....	32
3.1.2 Space Requirements.....	32
3.2 Electrical and Power Requirements.....	33
3.2.1 Electrical Requirements.....	33
3.2.2 Power Requirements.....	33
3.2.3 Plasma Source Power Requirements.....	33
3.3 Options for Non-Conforming Sites.....	34
3.4 Air Requirements.....	35
3.5 Plasma Source Requirements.....	36
INSTALLATION.....	37

TABLE OF CONTENTS

4.1 Before You Begin	38
4.2 Installation Tools and Items	39
4.3 Move the Pallet	40
4.4 Unpack the Machine Crate	41
4.4.1 Open the Crate	41
4.4.2 Unpack the Machine	41
4.5 Remove Frame from Crate	42
4.5.1 Lifting Frame	42
4.5.2 Installing Legs and Righting Machine	43
4.6 Level the Machine Frame and Water Table	45
4.6.1 Level the Machine Frame	45
4.6.2 Level the Water Table	45
4.7 Install Side Panels and Cable Carriers	47
4.8 Attach the Cable Guard	49
4.9 Install Plasma Torch and Lead	50
4.10 Make Electrical Connections	52
4.11 Install Machine Arm and Connect Controller	53
4.12 Fill Water Table	55
4.13 Verify the Installation	56
4.13.1 Check Machine Breakers	56
4.13.2 Power on the Machine	56
4.13.3 Verify Axes Function	57
4.13.4 Check Belt Tension	58
4.13.5 Testing Plasma Source Control	58
4.13.6 Testing Ohmic Probing	58
4.13.7 Testing Torch Breakaway	59
4.13.8 Power off the Machine	59
4.14 Set Torch Home Position (G30)	61
4.15 Calibrate Initial Height Sensing	62
4.15.1 Set Ohmic Probing Sensitivity	62
4.15.2 Set Physical Touch Switch Trigger Depth	62
4.16 Set Up the PathPilot Controller	63
4.16.1 Specify the Date and Time	63
4.16.2 Specify the Keyboard Language	63
4.16.3 Configure the Optional Touch Screen Kit	63
4.16.4 Update PathPilot	63
SYSTEM BASICS	65
5.1 System Reference	66
5.1.1 Water Table	66
5.1.2 Z Torch Head	66
<i>Ohmic Touchoff</i>	66
<i>Physical Touch Switch</i>	66
5.2 Basic Controls Reference	68
5.2.1 Machine Controls	68

5.2.2 PathPilot Interface	68
5.3 Connectors Reference	69
5.4 Plasma Source Control	70
PATHPILOT INTERFACE OVERVIEW	71
6.1 About PathPilot	72
6.2 Notebook Section	73
6.2.1 Main Tab	74
6.2.2 File Tab	75
6.2.3 Settings Tab	76
6.2.4 Offsets Tab	77
6.2.5 Conversational Tab	78
6.2.6 Status Tab	79
6.3 Persistent Controls	80
6.3.1 Program Control Area	81
6.3.2 Position Status Area	82
6.3.3 Manual Control Area	83
6.4 Keyboard Shortcuts	84
6.5 Manage PathPilot Versions	85
6.5.1 Download and Install an Update File from the Controller	85
6.5.2 Install an Update File from a USB Drive	85
6.5.3 Install a Previous Version of an Update File	86
PATHPILOT TOOLS AND FEATURES	87
7.1 Create and Load G-Code Files	88
7.1.1 Load G-Code	88
<i>Transfer Files to and From the Controller</i>	<i>88</i>
<i>Preview G-Code Files</i>	<i>88</i>
<i>Access Recent G-Code Files</i>	<i>89</i>
<i>Close the Current Program</i>	<i>89</i>
7.1.2 Edit G-Code with a Text Editor	89
7.1.3 Read G-Code	89
<i>Expand the G-Code Tab</i>	<i>89</i>
<i>About the G-Code Tab</i>	<i>90</i>
<i>Search in the Code</i>	<i>90</i>
<i>Set a New Start Line</i>	<i>91</i>
<i>Lead-In Moves</i>	<i>91</i>
<i>Change the View of the Tool Path Display</i>	<i>91</i>
<i>About the Tool Path Display</i>	<i>92</i>
7.1.4 Import a DXF File	92
<i>Working with Layers and Shapes</i>	<i>93</i>
<i>Change the Layer or Shape Cut Order</i>	<i>93</i>
<i>Adjust the Tree View Window</i>	<i>93</i>
<i>Working in the Preview Window</i>	<i>93</i>
<i>Create and Add Shape Library Templates</i>	<i>93</i>

TABLE OF CONTENTS

Create a Template.....	93
Add a Template.....	94
7.2 Machine Settings and Accessories.....	96
7.2.1 Enable an Internet Connection.....	96
7.2.2 Enable Automatic Updates.....	96
7.2.3 Change the Network Name.....	97
7.2.4 Change the Screen Orientation.....	97
<i>About Portrait Screen Layout.....</i>	<i>97</i>
7.2.5 Set Torch Touch Trigger Depth.....	98
7.2.6 Set Minimum Touchoff Spacing.....	99
7.2.7 Enable Ohmic Touchoff.....	99
7.2.8 Enable Torch Height Control.....	99
7.2.9 Enable Arc-Ok Checking.....	99
7.2.10 Disable Hard Stop Referencing.....	100
7.2.11 Limit G30 Moves.....	100
<i>About G30.....</i>	<i>100</i>
7.2.12 Enable the On-Screen Keyboard.....	100
<i>About Soft Keyboards.....</i>	<i>101</i>
7.2.13 Enable the USB M-Code I/O Interface Kit.....	101
7.2.14 Enable Tooltips.....	101
7.2.15 Use a USB Camera.....	101
<i>About USB Cameras.....</i>	<i>102</i>
<i>Manual Recording.....</i>	<i>102</i>
<i>Automatic E-Stop Loop Recording ("Dashcam").....</i>	<i>103</i>
<i>Review Video and Image Files.....</i>	<i>103</i>
File Naming Convention.....	103
G-Code Commands.....	103
File Naming Conventions.....	103
Use M01 to Take Pictures.....	104
7.3 Set Up G-Code Programs.....	105
7.3.1 Set Work Offsets.....	105
<i>About Work Offsets.....</i>	<i>105</i>
7.3.2 View Work Offsets.....	105
7.3.3 View Available G-Code Modes.....	106
7.4 Run G-Code Programs.....	107
7.4.1 Bring the Machine Out of Reset.....	107
<i>About Reset Mode.....</i>	<i>107</i>
7.4.2 View the Active Axis to Jog.....	107
7.4.3 Jog the Machine.....	107
<i>Jog in Continuous Velocity Mode.....</i>	<i>108</i>
About Continuous Velocity Jogging.....	108
<i>Jog in Step Mode.....</i>	<i>108</i>
About Step Jogging.....	108
7.4.4 View the Current Machine Position.....	108
7.4.5 Reference the Machine.....	108

<i>About Referencing</i>	109
7.4.6 Start a Program.....	109
<i>About Cycle Start</i>	109
<i>Cycle Start Reference</i>	109
7.4.7 Stop Machine Motion.....	109
7.4.8 View the Active G-Code Modes.....	109
7.4.9 View the Distance to Go.....	110
<i>About Distance to Go</i>	110
7.5 Control G-Code Programs	111
7.5.1 Use the Feed Hold Function.....	111
<i>About Feed Hold</i>	111
7.5.2 Use the Feed Rate Override Function.....	111
<i>About Feed Rate Override</i>	111
7.5.3 Use M01 Break Mode.....	112
<i>About M01 Break</i>	112
7.5.4 Use the Maxvel Override Function.....	112
<i>About Maxvel Override</i>	112
7.5.5 Use Single Block Mode.....	112
<i>About Single Block</i>	112
7.5.6 Use the Voltage Override Function.....	112
7.5.7 Change the Tool Number.....	113
<i>About M6 G43</i>	113
7.5.8 Use a G30 Position.....	113
<i>About G30</i>	113
7.5.9 Manually Enter Commands.....	113
<i>About the MDI Line DRO Field</i>	114
<i>Admin Commands Reference</i>	114
7.5.10 Copy Recently Entered Commands.....	115
7.5.11 Use the AutoFS Material Picker.....	115
<i>Writing G-Code with AutoFS</i>	115
<i>Using AutoFS</i>	115
<i>Creating Custom AutoFS Presets</i>	115
<i>Deleting Custom AutoFS Presets</i>	115
7.5.12 Use Cycle Counters (M30 and M99).....	116
<i>Monitor Cycle Counters</i>	116
<i>Change Cycle Counter Values</i>	116
7.6 System File Management	117
7.6.1 Manage System Files.....	117
7.6.2 Create Backup Files.....	117
<i>About Backup Files</i>	118
7.6.3 Restore Backup Files.....	118
7.6.4 Import and Export the Tool Table.....	119
<i>Import a .csv File</i>	119
<i>Export the Tool Table as a .csv File</i>	119

TABLE OF CONTENTS

BASIC OPERATIONS.....	121
8.1 Plasma Settings Reference.....	122
8.1.1 Touch Switch Trigger Depth.....	122
8.1.2 Minimum Touchoff Spacing.....	122
8.1.3 Enable Ohmic Touchoff.....	123
8.1.4 Enable THC.....	123
8.1.5 Enable Arc-Ok Checking.....	123
8.2 Create Programs with CAD/CAM.....	124
8.2.1 Converting DXF Files to G-Code.....	124
8.2.2 Using a Post-Processor.....	124
8.3 Select a Method for Initial Height Sensing (IHS).....	125
8.3.1 Physical Touch Switch.....	125
8.3.2 Ohmic Probing.....	125
8.4 Make Your First Cuts.....	126
8.4.1 Test G-Code.....	126
8.4.2 Code Breakdown.....	126
8.4.3 Run the Test Program.....	126
8.5 Use the AutoFS Material Picker.....	128
8.5.1 Writing G-Code with AutoFS.....	128
8.5.2 Using AutoFS.....	128
8.5.3 Creating Custom AutoFS Presets.....	128
8.5.4 Deleting Custom AutoFS Presets.....	128
PROGRAMMING.....	129
9.1 Before You Begin.....	130
9.2 Programming Overview.....	131
9.2.1 About G-Code Programming Language.....	131
9.2.2 G-Code Formatting Reference.....	131
<i>Line Numbers.....</i>	<i>131</i>
<i>Words.....</i>	<i>131</i>
<i>Letters.....</i>	<i>132</i>
<i>Values.....</i>	<i>132</i>
<i>Order of Execution.....</i>	<i>133</i>
<i>Modal Groups.....</i>	<i>134</i>
<i>Comments.....</i>	<i>135</i>
9.2.3 Supported G-Codes Reference.....	135
9.3 Programming G-Code.....	137
9.3.1 About the Examples Used.....	137
9.3.2 Rapid Linear Motion (G00).....	137
9.3.3 Linear Motion at Feed Rate (G01).....	138
9.3.4 Arc at Feed Rate (G02 and G03).....	138
<i>Radius Format Arc.....</i>	<i>138</i>
<i>Center Format Arc.....</i>	<i>139</i>
<i>Arc in XY Plane.....</i>	<i>139</i>

Arc in XZ Plane.....	139
Arc in YZ Plane.....	139
9.3.5 Dwell (G04).....	141
9.3.6 Set Offsets (G10).....	141
<i>Set Tool Table (G10 L1).....</i>	141
<i>Set Tool Table (G10 L10).....</i>	141
<i>Set Tool Table (G10 L11).....</i>	141
<i>Set Coordinate System (G10 L20).....</i>	141
9.3.7 Workpiece Probe (G15).....	142
9.3.8 Pierce (G16).....	142
9.3.9 Plane Selection (G17, G18, G19).....	142
9.3.10 Length Units (G20 and G21).....	142
9.3.11 Return to Predefined Position (G28 and G28.1).....	142
9.3.12 Return to Predefined Position (G30 and G30.1).....	142
9.3.13 Automatically Measure Tool Lengths with an ETS (G37 and G37.1).....	143
<i>Move to G37 Position Over ETS (G37.1).....</i>	143
<i>Move and Measure Tool Length (G37).....</i>	143
9.3.14 Straight Probe (G38.x).....	144
<i>Use the Straight Probe Command.....</i>	144
9.3.15 Cutter Compensation (G40, G41, G42).....	145
9.3.16 Dynamic Cutter Compensation (G41.1 and G42.1).....	145
9.3.17 Apply Tool Length Offset (G43).....	145
9.3.18 Engrave Sequential Serial Number (G47).....	145
9.3.19 Cancel Tool Length Compensation (G49).....	146
9.3.20 Absolute Coordinates (G53).....	146
9.3.21 Select Work Offset Coordinate System (G54 to G54.1 P500).....	146
9.3.22 Set Exact Path Control Mode (G61).....	147
9.3.23 Set Blended Path Control Mode (G64).....	147
9.3.24 Distance Mode (G90 and G91).....	147
9.3.25 Arc Distance Mode (G90.1 and G91.1).....	147
9.3.26 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	147
9.3.27 Feed Rate Mode (G93, G94, and G95).....	148
9.3.28 Spindle Control Mode (G96 and G97).....	148
9.4 Programming M-Code.....	149
9.4.1 Supported M-Codes Reference.....	149
9.4.2 Program Stop and Program End (M00, M01, M02, and M30).....	149
<i>Display Information and Capture Images During an M00 or M01 Break.....</i>	150
Display Information with Images.....	150
Display Information with Text.....	150
Capture Images with a USB Camera.....	150
9.4.3 Spindle Control (M03, M04, and M05).....	150
9.4.4 Tool Change (M06).....	150
9.4.5 Coolant Control (M07, M08, and M09).....	151
9.4.6 Override Control (M48 and M49).....	151
9.4.7 Feed Override Control (M50).....	151

TABLE OF CONTENTS

9.4.8 Spindle Speed Override Control (M51). 151

9.4.9 Set Current Tool Number (M61). 151

9.4.10 Set Output State (M64 and M65). 151

9.4.11 Wait on Input (M66). 152

9.4.12 USB Camera Control (M301, M302, M303). 152

9.4.13 Plasma Specific M-Codes. 152

9.5 Programming Input Codes. 154

9.5.1 Feed Rate (F). 154

9.5.2 Spindle Speed (S). 154

9.5.3 Change Tool Number (T). 154

9.6 Advanced Programming. 155

9.6.1 Parameters. 155

Parameters Reference. 155

 Parameter Syntax 155

 Parameter Scope. 155

 Behavior of Uninitialized Parameters. 155

 Parameter Mode. 155

 Persistence and Volatility. 155

 Intended Use. 155

Numbered Parameters Reference. 156

 Read-Only Parameters. 156

Subroutine Parameters Reference. 156

Named Parameters Reference. 156

9.6.2 Expressions. 157

Binary Operators Reference. 157

Functions Reference. 157

9.6.3 Subroutines. 158

Subroutines Reference. 158

Conditional Subroutines Reference. 159

 if/endif. 159

 if/elseif/else/endif. 159

Repeating Subroutines Reference. 159

Looping Subroutines Reference. 159

 do/while. 160

 while/endwhile. 160

MACHINE MAINTENANCE. 161

10.1 Maintenance Safety. 162

10.1.1 All Maintenance Procedures. 162

10.1.2 Swarf Maintenance Procedures. 162

10.2 Maintenance Schedules. 163

10.2.1 Daily. 163

10.2.2 Weekly. 163

10.2.3 Monthly. 163

10.2.4 Semi-Annually. 163

10.2.5 As Needed	163
10.2.6 Adjust Belt Tension	163
10.2.7 Adjust Rack and Pinion Preload	164
Y-Axis	164
X-Axis	164
TROUBLESHOOTING	167
11.1 Ohmic Probing	168
11.1.1 Ohmic Probe Does Not Trigger	168
11.1.2 Ohmic Probe Triggers Early or Between Cuts	169
11.2 Touch Off	171
11.2.1 The Torch Pierces Too High	171
11.2.2 The Torch Bends the Workpiece During Touch-off	171
11.3 Cutting	172
11.3.1 Torch Shuts Off During Cut	172
11.4 Voltage Feedback	173
11.4.1 No Voltage Reading in PathPilot	173
11.5 Torch Height Control	175
11.5.1 Torch Cuts Too High	175
11.5.2 Torch Drags on the Material When Cutting	176
11.6 Servo Drives	177
11.6.1 Servos Fault on Fast Direction Changes	177
DIAGRAMS AND PARTS LISTS	179
12.1 Base Machine	180
12.2 Frame Assembly	182
12.3 Water Table	185
12.4 Y-Axis Gantry Assembly	187
12.5 X-Axis Carriage Assembly	190
12.6 Z-Axis Lifter Assembly	192
12.7 Electrical Cabinet Layout	195
ELECTRICAL SCHEMATICS	197
13.1 230 Vac Power (Sheet 2)	198
13.2 24 Vdc Controls (Sheet 3)	199
13.3 Axis Drive Control (Sheet 4)	200
13.4 A-Axis Driver (Sheet 5)	201
13.5 Plasma Source Control (Sheet 6)	202
13.6 Machine Control Board (Sheet 7)	203
13.7 Torch Limit and Breakaway Switches (Sheet 8)	204
13.8 Accessory and Auxiliary Power (Sheet 9)	205
13.9 Grounds (Sheet 10)	206
13.10 Terminal Strips (Sheets 11-16)	207
13.10.1 TB1: AC Terminal Strip (Sheet 11)	207
13.10.2 TB1: AC Terminal Strip (Sheet 12)	208

TABLE OF CONTENTS

13.10.3 TB2: Control Terminal Strip (Sheet 13).....209

13.10.4 TB2: Control Terminal Strip (Sheet 14).....210

13.10.5 TB3: 75 Vdc Terminal Strip (Sheet 15).....211

13.10.6 TB4: Ground Terminal Strip (Sheet 16).....212

SAFETY

IN THIS SECTION, YOU'LL LEARN:

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



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

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

CONTENTS

1.1 Intended Use	16
1.2 Machine Standards	17
1.3 Safety Overview	18
1.4 Safety Messages	19
1.5 Safety Decals	20
1.6 Information Decals	22
1.7 Machine Safety	23

1: SAFETY

1.1 Intended Use

1.1 INTENDED USE

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

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

The intended use includes:

- Appropriate workholding, toolholding, tooling, systems, and machining parameters.
- Plasma cutting and engraving of ferrous and non-ferrous metals.

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

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

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

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

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

1.2 MACHINE STANDARDS

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

1.2.1 American National Safety Institute (ANSI)

- **ANSI B11.TR3-2000** *Risk Assessment and Risk Reduction — A Guideline to Estimate, Evaluate, and Reduce Risks Associated with Machine Tools*
- **ANSI Z49.1:2005** *Safety in Welding, Cutting, and Allied Processes*

1.2.2 Occupational Safety and Health Administration (OSHA)

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

1: SAFETY

1.3 Safety Overview

1.3 SAFETY OVERVIEW

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

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

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

1.4 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.

1.4.1 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.

1.4.2 Property Damage

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

1: SAFETY

1.5 Safety Decals

1.5 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.

1.5.1 On the Electrical Cabinet Door

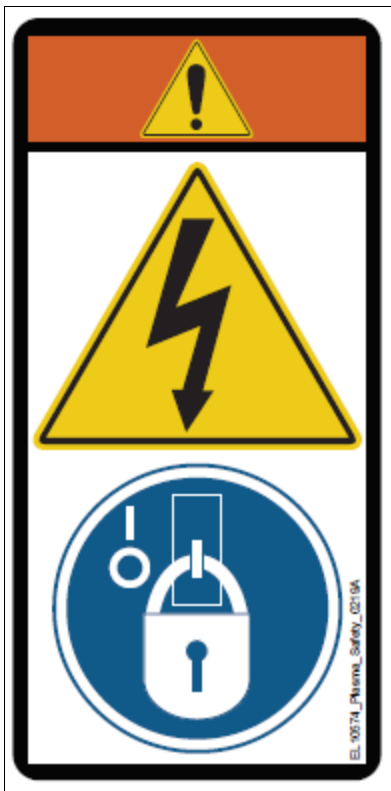


Figure 1-1: Example of a safety decal on the electrical cabinet door.

1. **WARNING! Electrocutation Hazard.** Points in the electrical cabinet contain high voltages, which can electrocute or shock you, causing death or serious injury. Even after the machine is powered off, electronic devices in the

electrical cabinet can retain dangerous electrical voltages. Use caution when servicing the machine inside the electrical cabinet.

2. **Lockout/Tagout.** Before servicing the machine, you must power off the machine and use an approved lockout/tagout device to secure the Main Disconnect switch in the **OFF** position. Points in the electrical cabinet contain high voltages, which can electrocute or shock you, causing death or serious injury.

1.5.2 On the Front of the Bed



Figure 1-2: Example of a safety decal on the front of the bed.

1. **WARNING! Arc Flash Hazard.** The plasma arc process produces very bright ultraviolet and infrared rays, which can injure your eyes and burn your skin. Always wear appropriate personal protective equipment to avoid damage to your eyes and skin.
2. **Don't Operate with an Active Implanted Cardiac Device.** If you wear an implanted cardiac device, you must consult with your doctor before operating this machine. The machine-generated electromagnetic fields could either interfere with or damage an implanted cardiac device.
3. **WARNING! Fire Hazard.** The machine is 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 does 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. **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+.
6. **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.

1.5.3 On the Gantry and the Plasma Head



Figure 1-3: Example of safety decals on the plasma head.

- **WARNING! Crush 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.

1: SAFETY

1.6 Information Decals

1.6 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.

1.6.1 Serial Number Plate

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



Figure 1-4: Example of the serial number plate on the side of the electrical cabinet.

1.7 MACHINE SAFETY



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

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

1.7.1 General Shop Safety	23
1.7.2 Operational Safety	23
1.7.3 Electrical Safety	24

1.7.1 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 safety eye protection with a shade number of 6 or greater when the torch is operating.
- ✓ Wear closed-toed safety shoes.
- ✓ Wear ear protection when you expect the machine or the machining processes to exceed safe exposure limits.
- ✓ Wear suitable work gloves when loading or unloading workpieces or stock that may be sharp or hot.
- ✓ Wear respiratory protection appropriate for the material being cut.
- ✓ Use welding blankets and screens to protect people and equipment outside of the machine's work area.
- ✓ Keep a clearly labeled fire extinguisher close by the machine's work area. The fire extinguisher should be rated for solid, combustible liquid, and electrical fires (NFPA Type ABC). The fire extinguisher should be inspected on a periodic basis as recommended by the manufacturer.
- ✓ Remain near the machine after use for at least one half

- ✓ hour to detect and extinguish any possible smoldering fires.
- ✓ 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.
- ✗ Never wear synthetic or synthetic-blend fabrics while operating the machine. Synthetic and synthetic-blend fabrics can melt causing severe injury. Only wear natural fiber fabrics.
- ✗ Keep flammable materials out of the machine's work area. Ensure that the walls, ceilings, and work surfaces of the workshop are suitably protected from sparks and hot materials.

1.7.2 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.
Guards may not stop all types of projectiles, like broken 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 or parts.
 - ✓ Clearing away chips, oil, or coolant.
 - ✓ Reaching into any part of the machine's motion envelope.

1: SAFETY

1.7 Machine Safety

- ✓ Removing protective shields or safeguards.
- ✓ Taking measurements.
- ✓ Doing any other action inside the machine's motion envelope.
- ✓ 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.
- ✓ 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 reach around a guard.
- ✗ Never allow the machine to run unattended.
- ✗ Never obstruct the Emergency Stop button or any other controls.
- ✗ Never allow untrained operators to install, operate, or maintain the machine.
- ✗ Never modify, defeat, or bypass safety devices or interlocks.
- ✗ Never machine abrasive, carcinogenic, explosive, flammable, radioactive, or toxic materials. Such materials include, but are not limited to:
 - ✗ Beryllium and its alloys
 - ✗ Ceramic
 - ✗ Fiberglass
 - ✗ G10 fiberglass laminate
 - ✗ Graphite
 - ✗ Lead and its alloys
 - ✗ Magnesium and its alloys

- ✗ Never allow swarf to accumulate on or within the machine.
- ✗ Never use flammable liquids (like alcohol, diesel fuel, or kerosene) in the machine's coolant system.
- ✗ Never use water, coolants without rust inhibitors, or straight cutting oil in the machine's coolant system.

Workholding

- ✓ Secure workpieces with appropriate workholding devices.
- ✓ Verify that the workpiece is adequately secured.
- ✓ Monitor cutting operations for "tip-ups" or warped workpieces that could collide with the machine.
- ✓ Remove cutoff workpieces and other large chips before starting the machine.
- ✗ Never leave tools, stock, or other loose items inside the machine.

1.7.3 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.
- ✓ 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.

TORMACH



ABOUT YOUR MACHINE

IN THIS SECTION, YOU'LL LEARN:

- About this machine's specifications.

CONTENTS

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

2: ABOUT YOUR MACHINE

2.1 Performance Expectations

2.1 PERFORMANCE EXPECTATIONS

2.1.1 Cutting Performance Reference

This machine is capable of cutting any metal that can be cut with a plasma torch at or near the recommended feeds and speeds. Verify that the programmed cuts do not exceed the maximum feed rate in X and Y.

- **Maximum Feed Rate** 1000 IPM (25.4 m/min)

2.1.2 Torch Height Control Reference

Success using torch height control depends largely on the voltage setting selected at run time, and its relation to cutting feed rate. Make sure that your material is flat enough that vertical movement of the torch would not exceed the maximum Z Torch Lifter feed rate.

- **Maximum Vertical Feed Rate** 120 IPM (3.0 m/min)



Tip! For a cut programmed at 140 IPM where the material rises 1" vertically over a 10" cut, the required vertical feed rate would be $(1/10) * 140 = 14$ IPM.

2.1.3 Resolution and Accuracy Reference

Accuracy is heavily influenced by the techniques that the machinist uses. A skilled machinist can deliver accuracy that exceeds the specified accuracy from the manufacturer; an inexperienced machinist may have difficulty delivering the specified accuracy. We can't predict operator accuracy, but the specified accuracy is an important reference point. The accuracy and overall quality of plasma-cut parts depends on plasma torch settings, air supply quality, workpiece material properties, toolpath parameters, and other dynamic factors. As such, the following accuracy specifications are for the machine's motion control system only.

- **Resolution** 0.0005" (.0127 mm)



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

- **Repeatability** $\pm 0.001"$ (± 0.0254 mm)
- **Positional Accuracy** ≤ 0.005 in./ft (130 micron / 300 mm)



Note: Positional accuracy takes into account items like backlash in the drive system and total composite error of the gear rack.

2.2 MACHINE SPECIFICATIONS

Travels	
X-Axis	48.5" (1232 mm)
Y-Axis	50" (1270 mm)
Z-Axis	3.9" (100 mm)
Maximum Feed Rate	
X- and Y-Axis	1000 IPM (25.4 m/min)
Z-Axis	120 IPM (3.0 m/min)
Power	
Primary Power Required	Single-Phase 230 Vac, 50/60 Hz
Recommended Circuit Amperage	20 A breaker
Table Specifications	
Table Size	51.2" x 51.2" (1300 mm x 1300 mm)
Table Type	Water Table
Weight Capacity (Excluding Water)	750 lb (340 kg)
Water Capacity	59 US gallons (223 liters)
Machine Specifications	
Gantry Clearance	7.0" (177 mm)
Machine Footprint	78" x 70" (1.95m m x 1.7 m)
Overall System Height	55" (1.4 m)
Typical System Weight	1232 lb (559 kg)
Linear Motion Components	
X and Y Axis Motor	Servo Driven
Z Axis Motor	NEMA 23 Stepper
Guideways	Precision Linear Guideway
Power Transmission	Planetary Gear (X Axis) 5mm HTD Belt (Y Axis)
Machine Construction	
Water Table	Welded Steel
Machine Base	Welded Steel
Gantry Bridge	Aluminum
Gantry Supports	Steel

2: ABOUT YOUR MACHINE

2.2 Machine Specifications

Controller	
Control System	PathPilot (v2.4.x or newer)
Resolution and Accuracy	
Resolution	0.0005" (.0127 mm)
Repeatability	±0.001" (±0.0254 mm)
Positional Accuracy	≤0.005 in./ft (130 micron / 300 mm)

SITE REQUIREMENTS

IN THIS SECTION, YOU'LL LEARN:

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



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

CONTENTS

3.1 General Site and Space Requirements	32
3.2 Electrical and Power Requirements	33
3.3 Options for Non-Conforming Sites	34
3.4 Air Requirements	35
3.5 Plasma Source Requirements	36



TORMACH®

3: SITE REQUIREMENTS

3.1 General Site and Space Requirements

3.1 GENERAL SITE AND SPACE REQUIREMENTS

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

3.1.1 Site Requirements

You must verify that the area:

- Allows for unrestricted access to machine controls.
- Conforms to the following:
 - **Primary Power Required** Single-Phase 230 Vac, 50/60 Hz
 - **Recommended Circuit Amperage** 20 A breaker



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

- Has a fire extinguisher within the work area.
- Is a dry, properly ventilated, and well-lit internal space.
- Provides for unobstructed machine motion and operation.

3.1.2 Space Requirements

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

- **Machine Size** 78" x 70" (1.95m x 1.7 m)
- **Machine Height** 55" (1.4 m)

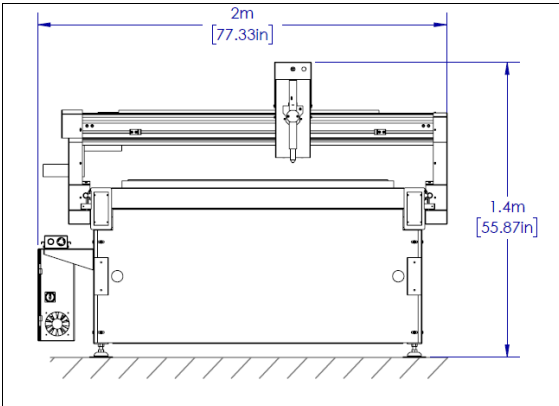


Figure 3-1: Dimensions of the machine itself, as viewed from the front.

- **Typical System Footprint** 89" x 67" (2.3 m x 1.7 m)

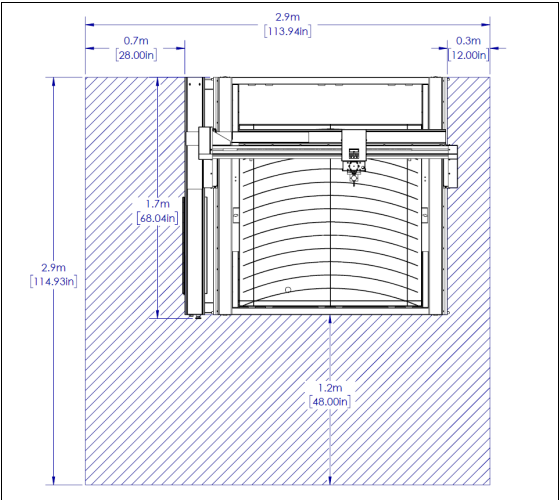


Figure 3-2: Dimensions of the machine and its required added space, as viewed from above.

3.2 ELECTRICAL AND POWER REQUIREMENTS

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

3.2.1 Electrical Requirements

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

- **Primary Power Required** Single-Phase 230 Vac, 50/60 Hz
- **Recommended Circuit Amperage** 20 A breaker

3.2.2 Power Requirements

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

- **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 6-20P plug, designed for use with a NEMA 6-20R receptacle.

3.2.3 Plasma Source Power Requirements

In addition to the requirements listed above for the machine itself, make sure that you have a separate circuit available to power your plasma source. For example, when using 220V single-phase power the Hypertherm Powermax 45 XP requires the following:

- **Primary Power Required** 200 Vac – 240 Vac
- **Recommended Circuit Amperage** 50 A
- **Plug Pattern** NEMA 6-50P

Check with the manufacturer documentation for the specific plasma source you are using before installation.

3: SITE REQUIREMENTS

3.3 Options for Non-Conforming Sites

3.3 OPTIONS FOR NON-CONFORMING SITES

For sites that don't conform to the specified "Electrical and Power Requirements" (on the previous page), 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, 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 performance or damage to electrical components, we recommend the Buck-Boost Transformer (PN 32554).

3.4 AIR REQUIREMENTS

You must verify that the site conforms to the following air supply requirements.

- **Air Pressure** 90-120 psi (620-825 kPa)
If the air supply is more than 120 psi (825 kPa), you must use a regulator.
- **Air Volume** At least 6 cfm at 90 psi. Check with your plasma source manufacturer in case your model requires more volume.
- **Dry Air** We recommend using a compressed air dryer, desiccator, or filter between the air compressor and the machine.

3: SITE REQUIREMENTS

3.5 Plasma Source Requirements

3.5 PLASMA SOURCE REQUIREMENTS

Tormach recommends either the Hypertherm Powermax 45 XP or the Powermax 65 plasma sources for use with the 1300PL. Plasma sources ordered directly from Tormach will include the correct CNC control interface and cables.

If you would like to use your own plasma source, the minimum requirements are as follows:

- Blowback style arc-start. The 1300PL is not recommended for use with high-frequency style arc-start units. The electrical noise spike generated by high-frequency arc starts can interfere with the axis control signals.
- An internal voltage divider outputting a 50:1 torch voltage signal.
- An input for plasma arc start compatible with dry contact closure to activate (the 1300PL uses an open relay for torch off and closed relay for torch on)
- (Optional but recommended) Arc-ok output. Closed contacts when arc is active.

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

4.1 Before You Begin	38
4.2 Installation Tools and Items	39
4.3 Move the Pallet	40
4.4 Unpack the Machine Crate	41
4.5 Remove Frame from Crate	42
4.6 Level the Machine Frame and Water Table	45
4.7 Install Side Panels and Cable Carriers	47
4.8 Attach the Cable Guard	49
4.9 Install Plasma Torch and Lead	50
4.10 Make Electrical Connections	52
4.11 Install Machine Arm and Connect Controller	53
4.12 Fill Water Table	55
4.13 Verify the Installation	56
4.14 Set Torch Home Position (G30)	61
4.15 Calibrate Initial Height Sensing	62
4.16 Set Up the PathPilot Controller	63



TORMACH®

4: INSTALLATION

4.1 Before You Begin

4.1 BEFORE YOU BEGIN

1. Inspect the item(s):

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

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

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

- Accessories (if applicable)
- Machine



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

4.2 INSTALLATION TOOLS AND ITEMS

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

- Tin snips
- Flat-blade screwdriver
- Pliers
- 3mm, 4mm, 5mm, 6mm Allen key
- Dead-blow hammer
- Utility knife
- Adjustable crescent wrench
- Pallet jack
- Engine hoist, forklift, or gantry crane
- Lifting sling (2200lb / 1000kg minimum rating)

4: INSTALLATION

4.3 Move the Pallet

4.3 MOVE THE PALLET

Tools and Items Required

- Pallet jack

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

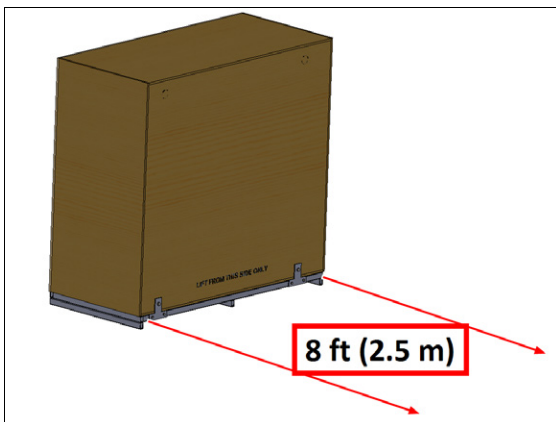
WARNING! Transportation and Lift Hazard: Before moving the machine, you must confirm that all persons are clear of the area below the machine. Qualified professionals must transport, lift, and move the machine. Moving parts can entangle, pinch, or cut you, causing death or serious injury.

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



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

One side of the crate is labeled **Lift from this side only**. Verify that there's at least 8 ft (2.5 m) of free space behind this side of the crate so that you can lay the machine down flat on the ground.



4.4 UNPACK THE MACHINE CRATE

Tools and Items Required

- Pallet jack
- Flat-blade screwdriver
- Pliers
- Tin snips
- Safety eyewear that meets ANSI Z87+
- Work gloves



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

area.

2. Remove the Hypertherm control cable and package of small parts from the crate.
3. Unstrap and remove the cable tray and cable tray cover.
4. Unstrap and remove the cable tray support brackets.
5. Unstrap and remove all four legs.
6. Remove the four side panels from underneath the machine frame.



CAUTION! Be careful not to hit the torch lifter head while removing panels from the crate.

7. Remove all plastic overwrap from the machine frame and water table except for the Y axis energy chain (it will be unwrapped later).

4.4.1 Open the Crate

1. Put on work gloves and eye protection.
2. Pry up the tabs holding the lid on to the sides of the crate. You may need to use pliers to fully straighten them so the lid can be removed.
3. With a helper, lift the lid off of the crate and set aside.
4. Remove the screws from the base of the crate on all four sides.
5. Straighten the tabs holding one of the panels on to the crate. The front panel is opposite the side of the crate stenciled "Lift from this side only".
6. Remove the front panel from the crate and set aside.
7. With a helper, lift the remaining three sides from the crate as a unit.
8. 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.

4.4.2 Unpack the Machine

1. Cut the strap holding the electrical cabinet to the crate and unwrap the plastic from it. Set it aside in a safe

4: INSTALLATION

4.5 Remove Frame from Crate

4.5 REMOVE FRAME FROM CRATE

This portion of the installation requires at least two people. Make sure that you have a partner before beginning the lifting procedure.

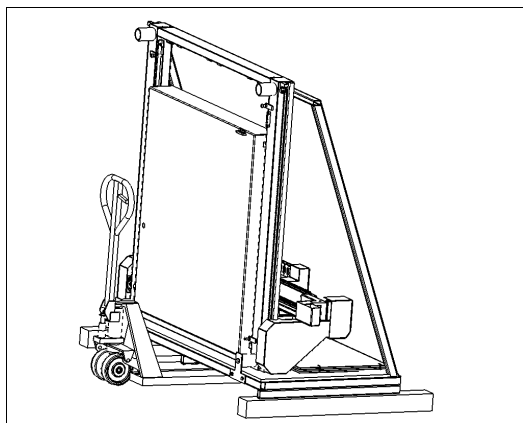
Tools and Items Required

- Engine hoist or gantry crane rated for 825 lb (375 kg)
- Pallet jack
- 4x4 lumber (4ft long)
- Lifting strap
- Pry bar
- Allen keys
- Safety eye-wear that meets ANSI Z87+
- Work gloves

CAUTION! Lifting a machine can be dangerous if you are unfamiliar with your equipment. Make sure to know the load ratings of your lifting equipment and hire a qualified rigging company if you are not confident in its safe operation.

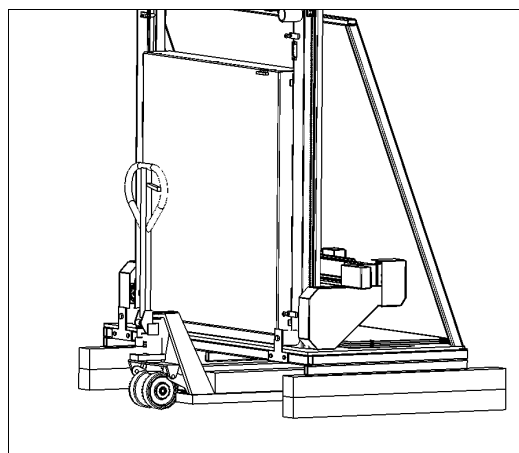
4.5.1 Lifting Frame

1. If using an engine hoist, perform the following steps to lift the crate for clearance underneath. If using a forklift or gantry, proceed to the next step.
 - a. Position a pallet jack under the machine and lift high enough to place a 4x4 under the side supports.



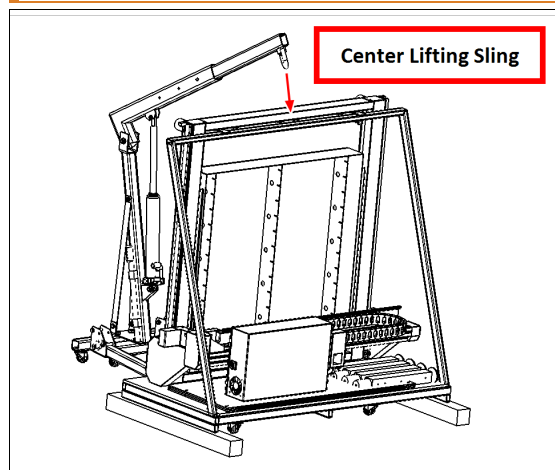
- b. Place two 4x4 pieces on the pallet jack legs and repeat the lift. Place another 4x4 under the side

supports.



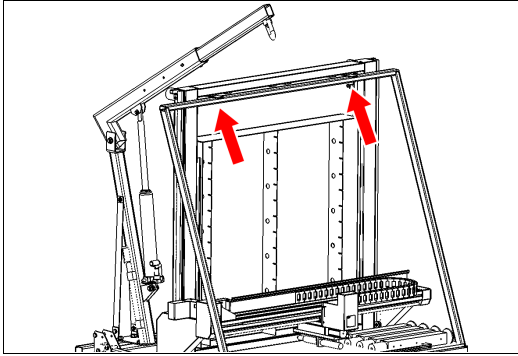
- c. Repeat the above steps until the crate clears the front legs of your engine hoist.
2. Attach a sling to the top frame rail of the plasma. Make sure the sling is positioned securely between two of the raised accessory mounting pads to prevent side-to-side slippage of the frame.

WARNING! Make sure that the lifting device you are using is rated for the weight of the machine frame - 825 lb (375 kg).

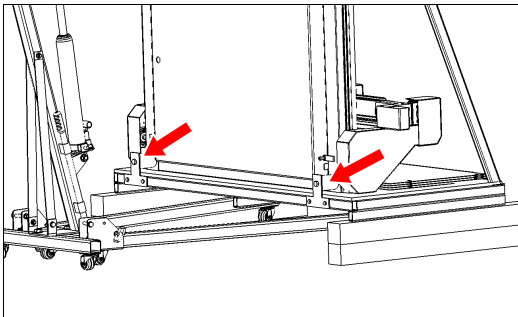


3. Position your engine hoist, crane or forklift above the sling and put light tension on it, just enough to hold the frame upright when it is released.

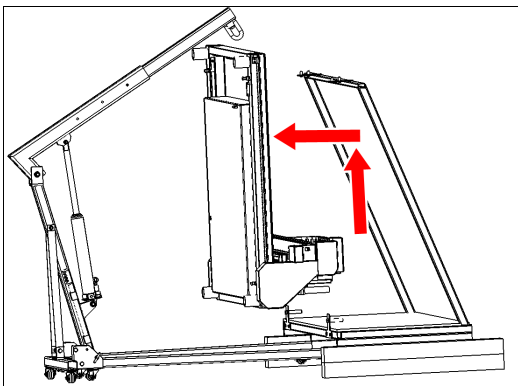
4. Remove the top two bolts holding the A-frame support of the crate to the machine frame.



5. Remove the two bottom bolts connecting the crate frame to the machine leg mounting pockets.

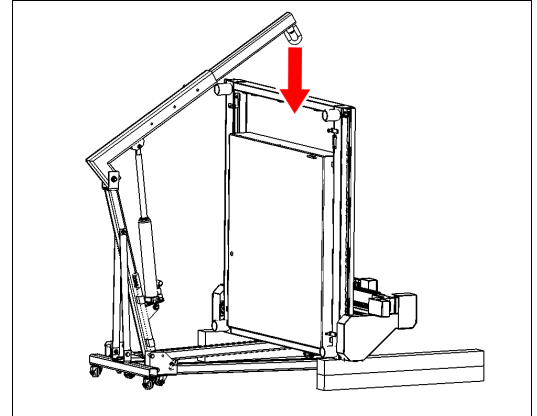


6. Use your lifting device to put enough upwards pressure on the machine that it will be fully supported. Use a pry bar to push the lower leg mounting pockets off of the crate mounting bosses.
7. Lift the machine frame enough to clear the crate and use a pallet jack to remove the crate from the area.

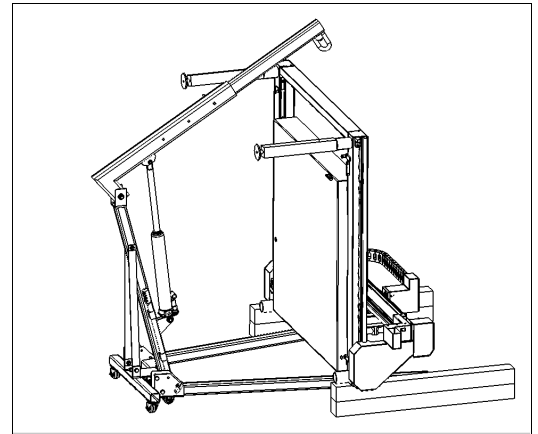


4.5.2 Installing Legs and Righting Machine

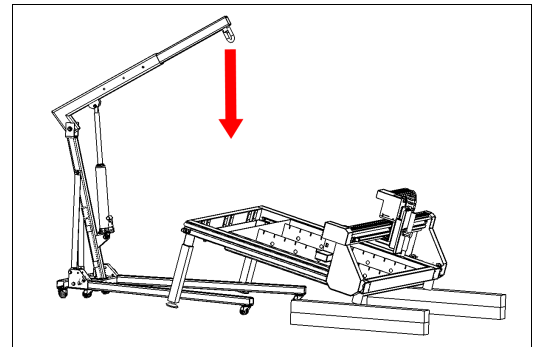
1. Lower the bottom end of the frame down on to 4x4 blocks to protect the paint and clear the engine hoist legs (if present).



2. Install the top two legs on the machine on the end supported by the lifting sling.



3. Using gantry crane, slowly lower the top side of the machine frame until the two legs are on the ground.

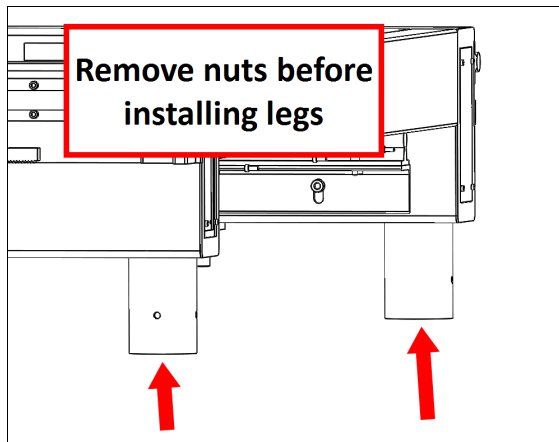


4. Move the lifting sling to the same position on the opposite end of the frame.

4: INSTALLATION

4.5 Remove Frame from Crate

5. Lift the remaining end of the frame off of the 4x4 blocks.
6. Remove the two nuts (used to bolt to the crate frame) from inside the leg mounting holes. Do not discard the socket head cap screws attaching the nuts, these are used to attach the last two legs.



7. Install the last two legs.

4.6 LEVEL THE MACHINE FRAME AND WATER TABLE

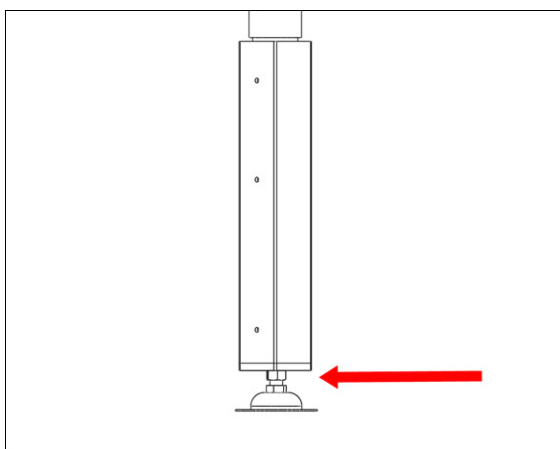
Tools and Items Required

- Carpenter's Level
- Crescent Wrench
- Allen Key

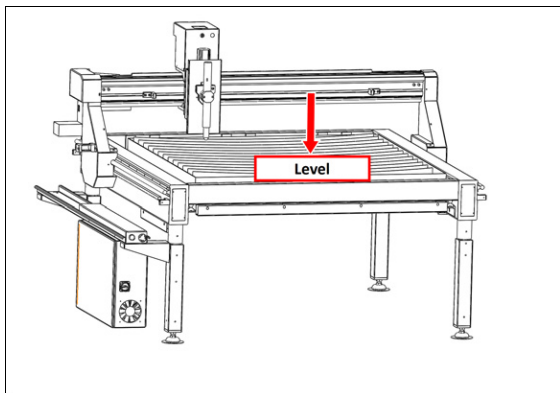
The water table can be leveled at any time, but it is easiest to do before the side panels have been installed.

4.6.1 Level the Machine Frame

1. Loosen the jam nuts on all machine feet

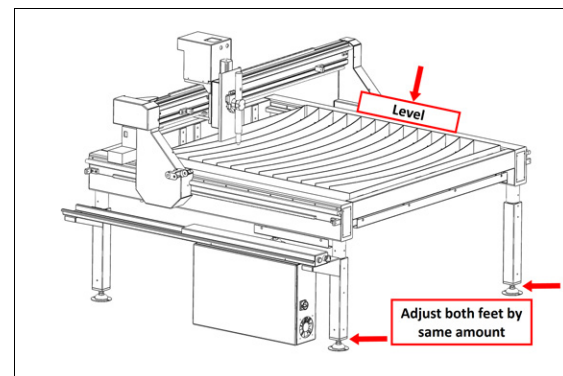


2. Place a carpenter's level across the front frame rail of the machine and level it side-to-side using the threaded feet.



3. Perform the same procedure on the rear of the machine, leveling it side-to-side.
4. Place the level on one of the side rails of the machine frame and level it front to rear.
You must adjust both front or rear feet the same number of turns while doing this to avoid losing side-to-side level

from the step above.

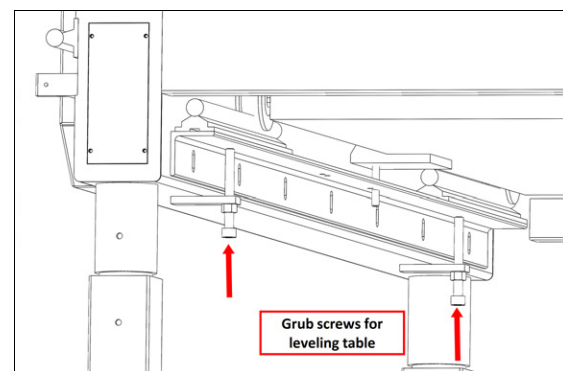


5. Tighten all the jam nuts for the machine feet.

4.6.2 Level the Water Table

Note: If the water tank ball valve is not already installed, apply sealant tape to the water tank drain threads and install the ball valve onto the water tank.

1. Examine the level on the water table to determine if you must make adjustments. The water table on the 1300PL is supported on two adjustable rails, one on each side of the table. The frame has grub screws to adjust the height of these rails, leveling the water table to the machine frame.

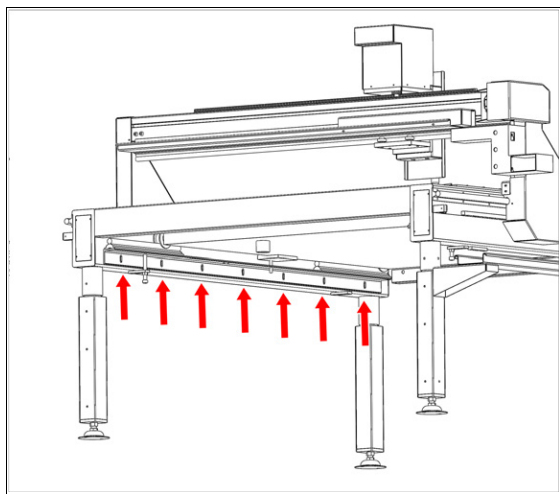


2. Depending on what you determined in Step 1, do one of the following:
 - If the water table is level, proceed to the next section.
 - If the water table isn't level, go to Step 3.

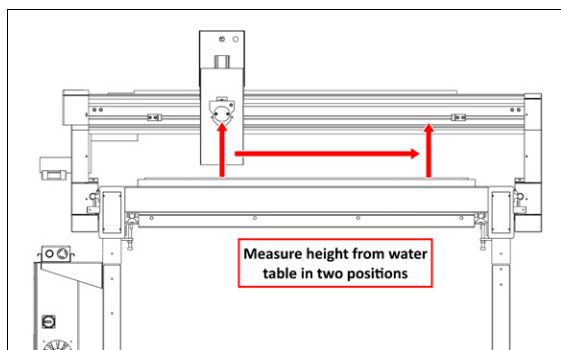
4: INSTALLATION

4.6 Level the Machine Frame and Water Table

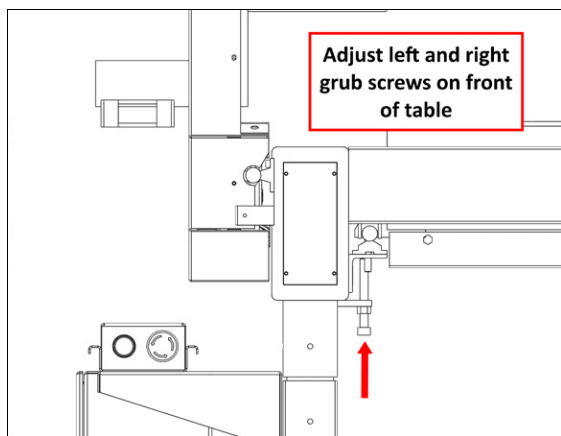
- Loosen the socket head cap screws holding the water table support rails to the machine frame, just enough that the rail can move vertically.



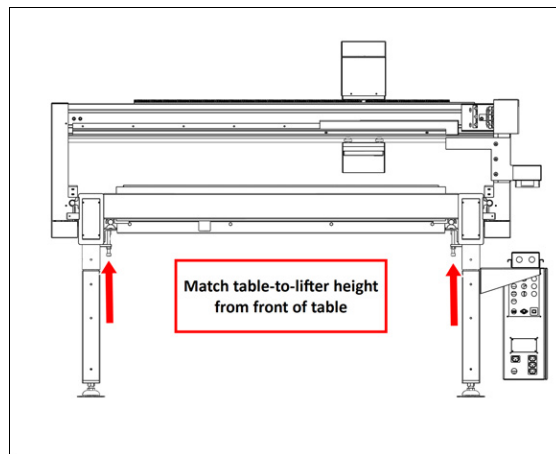
- By hand, move the torch carriage to the front left and right corners of the water table. Measure the vertical distance from the torch holder to the water table slats.



- Using the two grub screws underneath the machine frame, lift the water table rails so that the front two corners are level.



- Move the torch carriage to the rear of the water table and repeat the procedure for the rear grub screws.



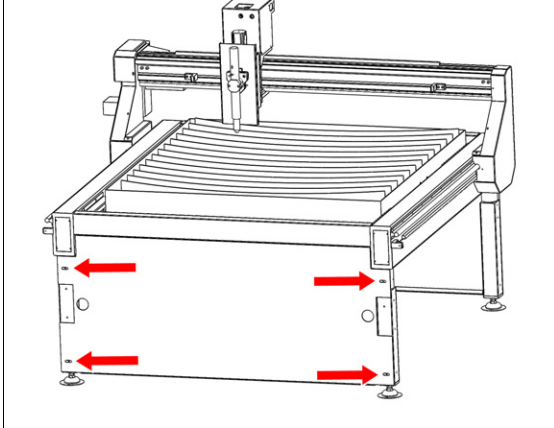
- Re-check the front corners and iterate the leveling process if needed until your table-to-lifter height matches in all four spots.

Note: Small variations in level will be compensated for by torch height control. A table leveled to within 1/4" (6 mm) will perform well with THC.

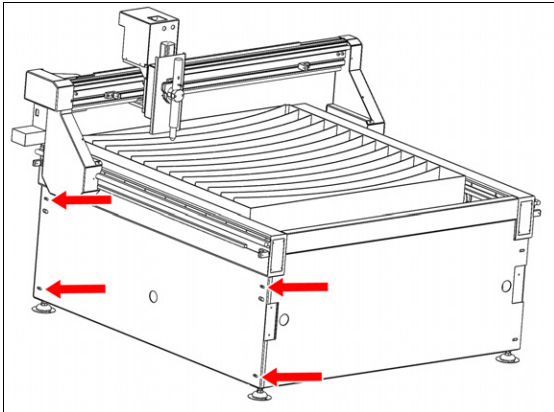
- Tighten all the socket head cap screws holding the water table rails to the frame.

4.7 INSTALL SIDE PANELS AND CABLE CARRIERS

1. Install the front and rear panels using the M8 button head cap screws and washers from the small parts package. To prevent vibration, use one of the included flat washers and then one lock washer for the top two mounting screws on each of the panels.

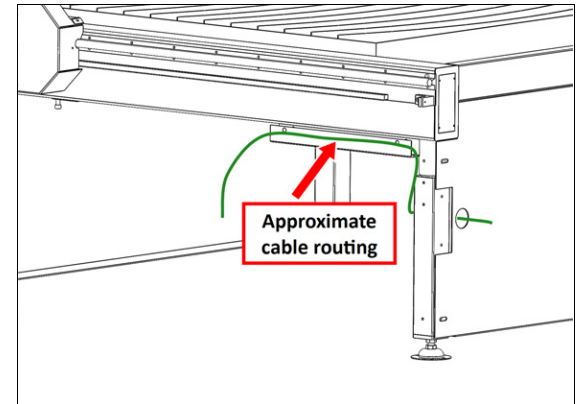


2. Install the left side panel (viewed from the front), with the flanges overlapping the front and rear panels. The right side panel will be left off for now to allow access to the bottom of the machine.

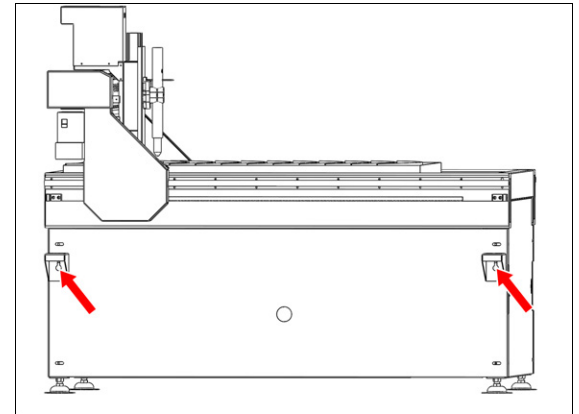


Note that this side panel has a small cable carrier on the inside of the frame to carry cables from the machine

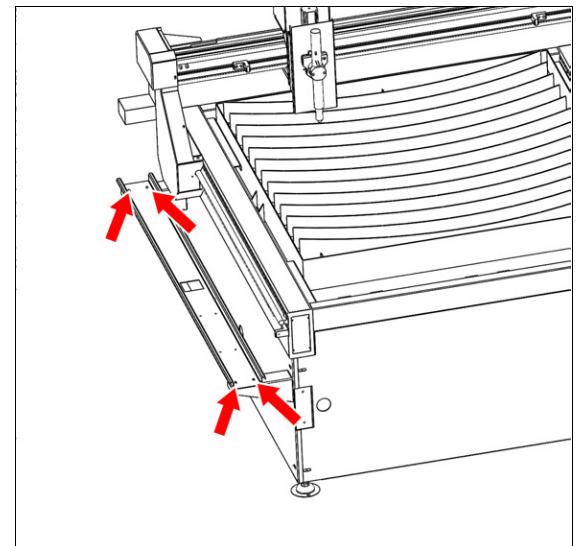
arm to the electrical cabinet.



3. Hang the cable track mounts on the two M8 button head mounting screws.



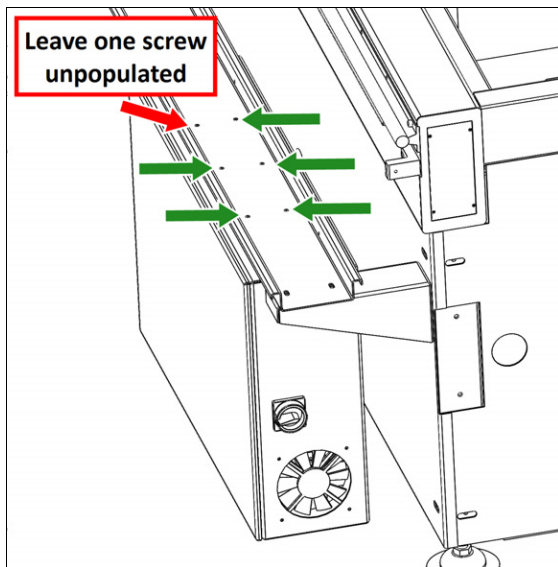
4. Remove the top cover from the cable track (4x M3 screws). It will be reinstalled after hanging the electrical cabinet.
5. Install the cable track to the mounts with four M6 button head cap screws.



4: INSTALLATION

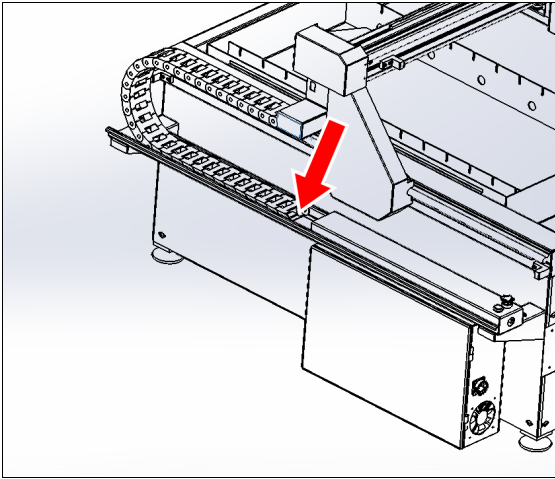
4.7 Install Side Panels and Cable Carriers

6. Hang the electrical cabinet from five M6 screws, leaving the one nearest the pass-through hole for electrical wires off (this is where a ground wire will be attached).



4.8 ATTACH THE CABLE GUARD

1. Unwrap the flexible black cable guard from where it is bundled on the side of the gantry and lay it out onto the cable track above the electrical panel.
2. Pass the bundle of connectors from the flexible cable guard down through the rectangular cutout in the cable track.
3. Attach the cable guard to the cable track using 4 M6 socket head cap screws.

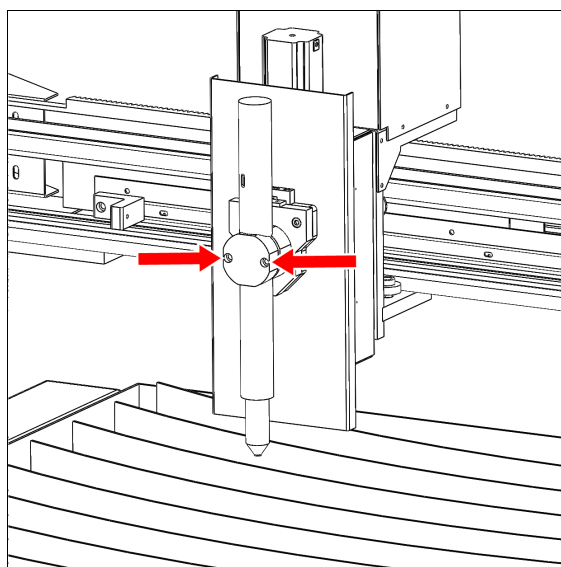


4: INSTALLATION

4.9 Install Plasma Torch and Lead

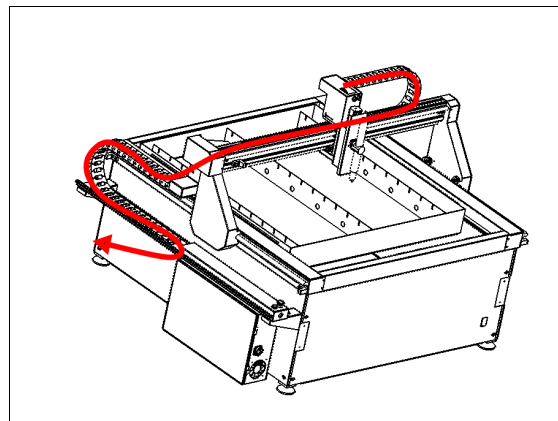
4.9 INSTALL PLASMA TORCH AND LEAD

1. Install the cable retention loop into the base plate.
2. Locate the ohmic torch wire inside the X-axis motor cover. Pull it out and let it hang down for now.
3. To prepare for the installation of the torch lead, remove all of the side-to-side covers from one side of the flexible cable guard using a flat-blade screwdriver.
4. Remove the plasma power supply unit from its packaging.
5. Put the plasma power supply at the back left of the machine. Then, locate the torch lead.
6. Remove the brass gear from the torch body. Depending on which torch you have, do one of the following:
 - Unthread the body and slide the gear out of the housing.
 - Remove the two screws securing the gear.
7. Using the manufacturer's instructions, install the torch consumables.
8. Install the plasma torch into the breakaway torch holder using the two socket head screws. Only tighten these screws enough to securely hold the torch, do not overtighten.

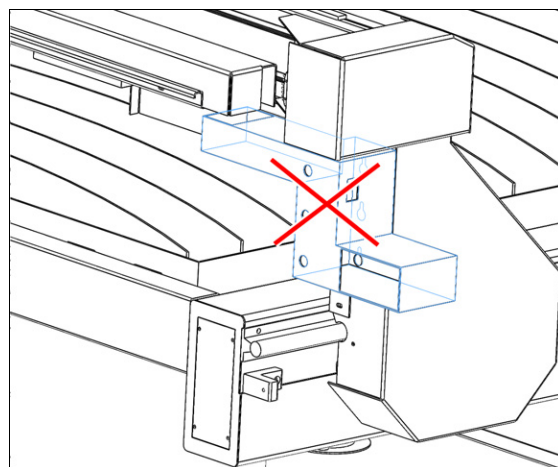


Tip! We recommend positioning the torch nose about 1" below the Z-axis cover plate to start.

9. Attach the ohmic probing spade connector to your torch cap.
10. Install plasma torch lead into the cable guard from the X carriage to the electrical cabinet and pass it through the rectangular access hole to bottom of track.



Note: Do not try to pass the torch lead through the stepped section between the Y and X axis cable guards. Simply run the torch lead along the outside.



11. Plug the torch leads into the plasma power supply.
12. Plug in the ground cable, and route the loose end of the cable under the back of the machine and up to the water table.

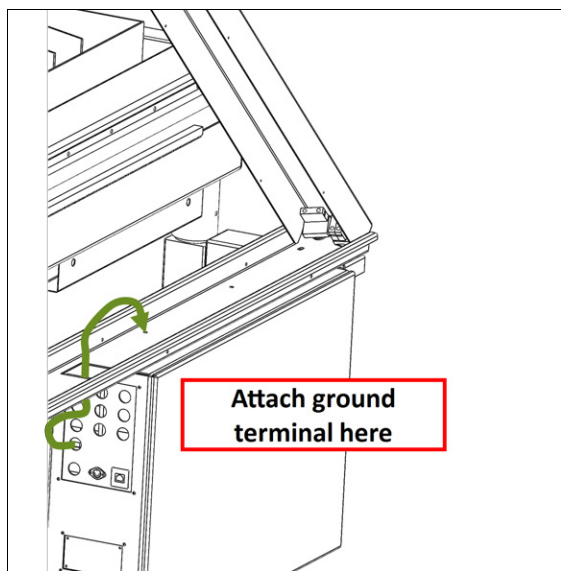
13. Connect the included Hypertherm control cable to the electrical cabinet and plasma source (this cable is a 14-pin CPC connector on one end and a 7-pin threaded -body connector on the other). For more information on this cable see "Plasma Source Control" (page 70).
14. Reinstall the cable guard covers that you removed in Step 3.

4: INSTALLATION

4.10 Make Electrical Connections

4.10 MAKE ELECTRICAL CONNECTIONS

1. Locate the cable cover with the integrated E-Stop and Reset button. The cable for the Reset button and E-Stop contains a green and yellow ground lead with a ring terminal on the end. Attach this ground lead for the reset panel to the fourth M6 mounting screw on the electrical cabinet that you left unpopulated earlier.



2. Pass the connector for the reset panel down through cable track.
3. Install the reset panel / cable cover using four M4 socket head cap screws.

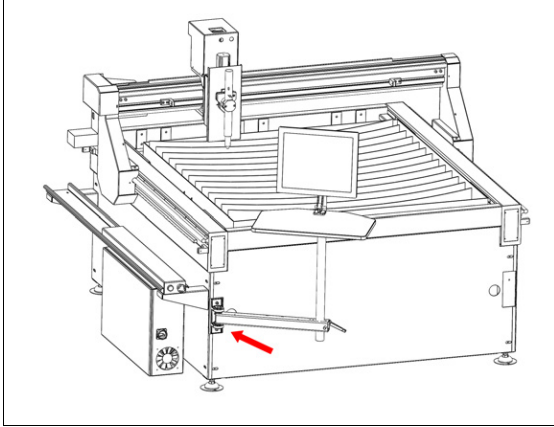


Note: The two inboard screws are slightly longer.

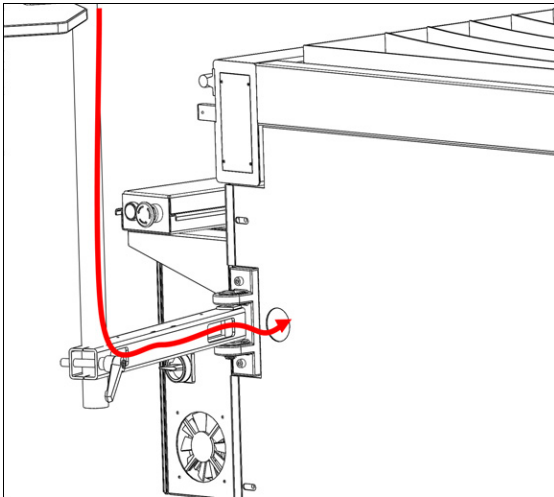
4. Plug in all connectors at rear of cabinet, following the "Connectors Reference" (page 69) guide in the "System Basics" (page 65) section.
5. Pass the ground wire through the hole in the side panel, and then connect it to the ground block on the underside of the water table.

4.11 INSTALL MACHINE ARM AND CONNECT CONTROLLER

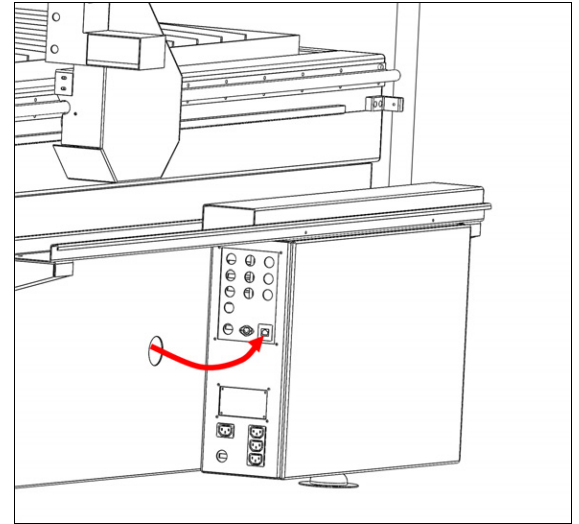
1. Bolt your machine arm to the mounting pad on either the left or right hand leg of the machine.



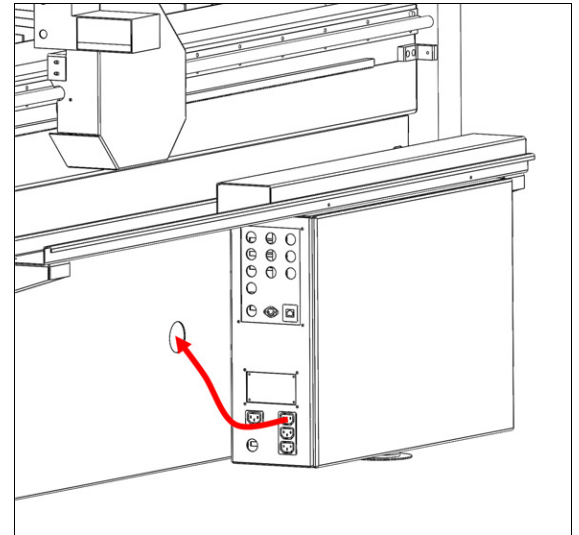
2. Install the monitor, keyboard tray and controller mount on the machine arm.
3. Mount the controller PC to the back of the monitor using the PathPilot Controller VESA Mount (PN 50382).
4. Connect a keyboard and mouse to the controller PC.
5. Run the Ethernet cable included in your machine owner's kit down from the controller and through the hole in the front panel of the plasma table.



6. Connect the Ethernet cable to the port on the rear of the electrical cabinet.



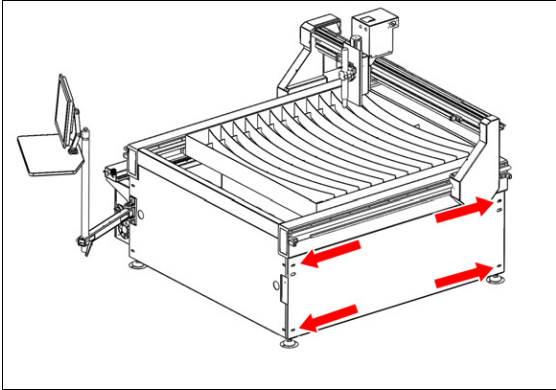
7. Run a power cable from the rear of the electrical cabinet to the power supply for the controller PC in the same way. Use the block of three IEC ports designated for Accessory Power in "Connectors Reference" (page 69). Repeat for the display's power cable.



4: INSTALLATION

4.11 Install Machine Arm and Connect Controller

8. Now that all connections have been made inside the machine frame, the right side panel can be installed. Before installing the panel, check that the drain valve on the water table is fastened securely and in the closed position.



9. Install all end panels on the frame rails.

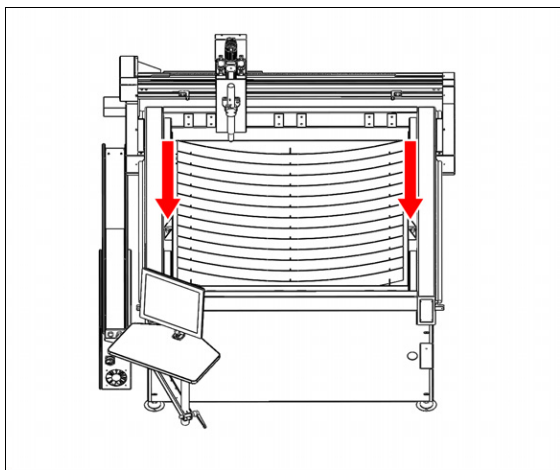
4.12 FILL WATER TABLE

Tools and Items Required

- Hose or bucket
- Plasma cutting fluid (rust inhibitor such as GreenCut or similar)

A properly filled water table is critical for safe operation of the 1300PL. The water table traps a majority of the metal particulates released in the cutting process and keeping it filled is important for maintaining good air quality in your shop space.

1. Ensure that the drain valve on the bottom of the water table is in the closed position.
2. Check that the two lock bolts that hold the table in place on the front-to-back tracks are in place.



3. Using a hose or buckets, fill the table with tap water until the level is about 1" below the top of the slats.
4. Add cutting fluid to the water to inhibit corrosion. Use the ratio recommended by the cutting fluid manufacturer. For reference, full water table capacity on the 1300PL is 59 US gallons (223 liters).
5. Top the water table off to 0.5" below the top of the slats after adding cutting fluid if desired.

4: INSTALLATION

4.13 Verify the Installation

4.13 VERIFY THE INSTALLATION

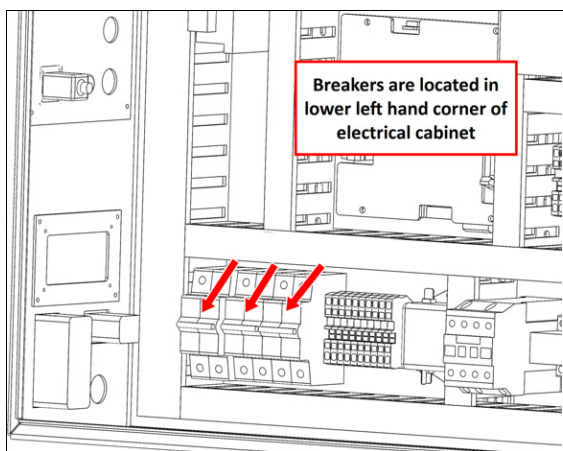
To properly validate the core installation of your machine, you must understand how to power on and off the machine and use the controls.

Complete the following steps in the order listed:

4.13.1 Check Machine Breakers.....	56
4.13.2 Power on the Machine.....	56
4.13.3 Verify Axes Function.....	57
4.13.4 Check Belt Tension.....	58
4.13.5 Testing Plasma Source Control.....	58
4.13.6 Testing Ohmic Probing.....	58
4.13.7 Testing Torch Breakaway.....	59
4.13.8 Power off the Machine.....	59

4.13.1 Check Machine Breakers

Check that all breakers in the electrical cabinet are switched on (up position).



4.13.2 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** 20 A breaker
2. Connect the machine's mains power cable to the verified electrical service.
3. Find the Main Disconnect switch, and then remove the hang tag.

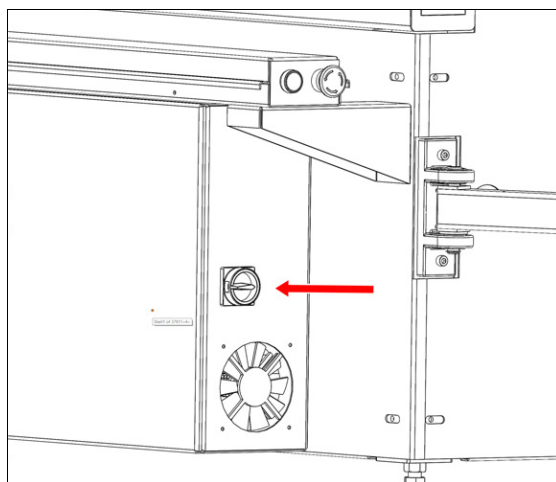


Figure 4-1: The Main Disconnect switch on the side of the machine's electrical cabinet.

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

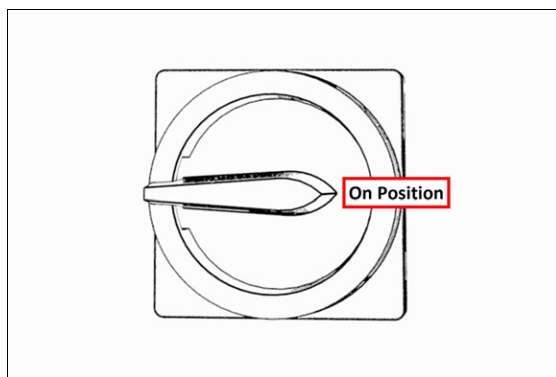


Figure 4-2: Example of the Main Disconnect switch in the On position.

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

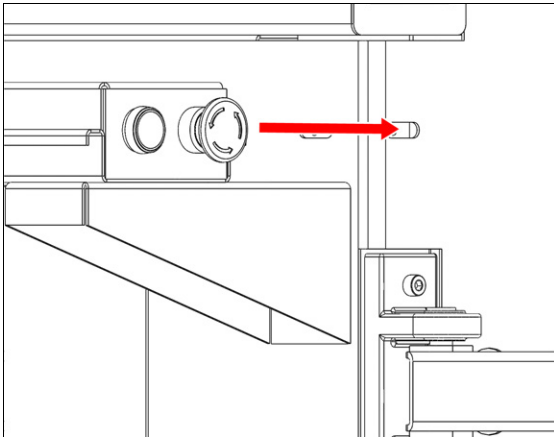


Figure 4-3: Emergency Stop button.

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

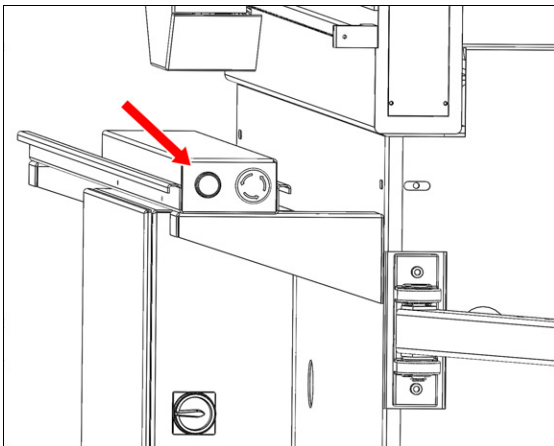


Figure 4-4: Reset button.

The axis drives are now powered on.

10. 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.

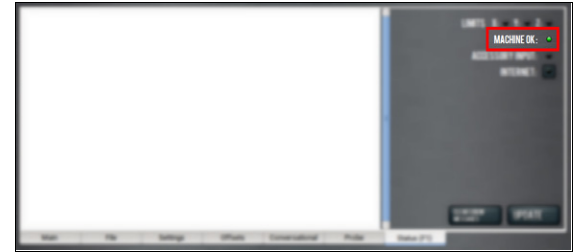


Figure 4-5: Machine OK light on the Status tab.

11. Select **Reset**.



Figure 4-6: Reset button.

This initializes the connection between the machine and controller.

4.13.3 Verify Axes Function

You must confirm that the axes correctly operate.

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



Figure 4-7: 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 **Right Arrow** key and then the **Left Arrow** key.
The torch carriage moves right (X+), then left (X-).
 - Select the **Up Arrow** key and then the **Down Arrow** key.
The gantry moves toward the back end of the table (Y+), then toward the front (Y-).
 - Select the **Page Down** key and then the **Page Up** key.
The torch lifter moves down (Z-), then up (Z+).

4: INSTALLATION

4.13 Verify the Installation

- If applicable, verify the optional Jog Shuttle:
 - Press any axis button on the Jog Shuttle — X, Y, Z, or A — to select an axis.

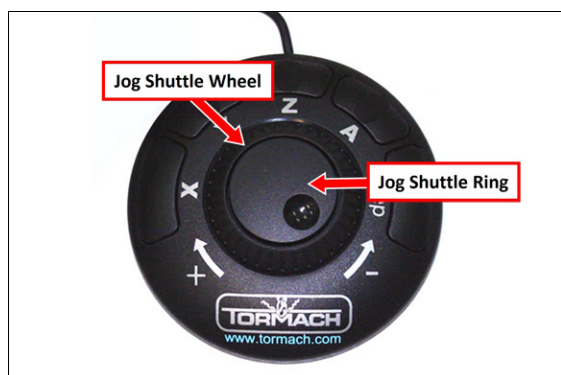


Figure 4-8: Functions on the optional Jog Shuttle.

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



Figure 4-9: 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.

4.13.4 Check Belt Tension

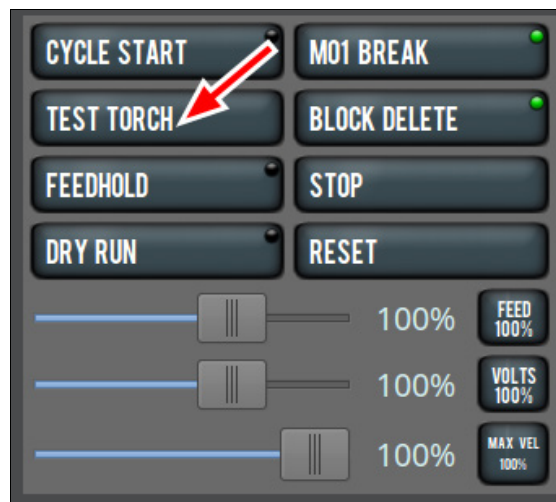
Since brand new drive belts have a tendency to stretch slightly after first use, your Y-axis drive belts might need re-tensioning. If you notice that the Y axis gantry seems to have excessive backlash or that the servo is struggling to hold a consistent position, follow the maintenance procedure for "Adjust Belt Tension" (page 163)

If Y-axis gantry movement seems normal and there is no noticeable backlash belt tension adjustment can be deferred until the time defined in the "Maintenance Schedules" (page 163).

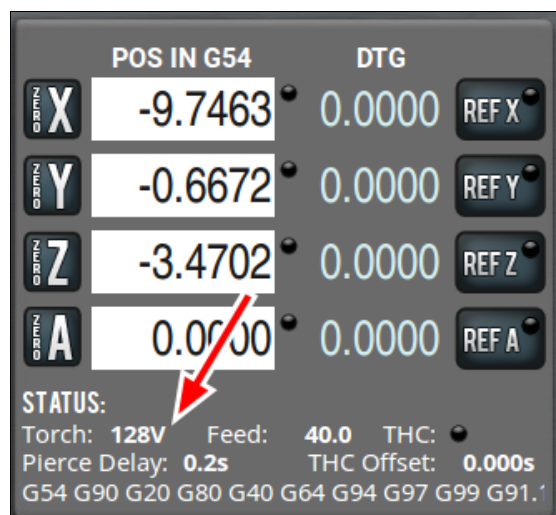
4.13.5 Testing Plasma Source Control

- Using the "Connectors Reference" (page 69) verify that the Hyperthem control cable is connected to the correct spot on the rear of your electrical cabinet.
- Jog the torch to a location several inches above the water table so that you can safely test the arc.

- Activate the plasma source and create a pilot arc by pressing the **TEST TORCH** button in PathPilot.



- Verify that your plasma source activates and creates a pilot arc (you may have to press the button twice to bypass the "warning puff" function on some plasma sources).
- While the pilot arc is active, watch the "Torch Voltage" readout at the bottom of your control screen.



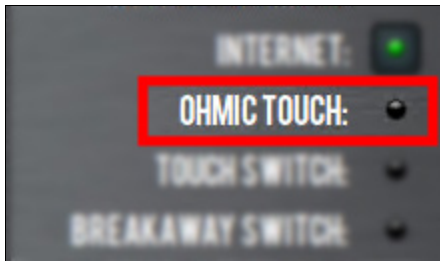
You should see an open-circuit voltage of above 100V. If you do not see a voltage reading, follow the "Voltage Feedback" (page 173) section of the Troubleshooting guide.

4.13.6 Testing Ohmic Probing

- Make sure that the ohmic probing spade connector is connected to your torch cap.

Note: You must be using shielded consumables where the torch cap extends below the nozzle. If the nozzle contacts the workpiece before the torch cap, ohmic probing will not work.

2. Switch to the Status tab on your PathPilot controller and locate the **OHMIC TOUCH** led. It should be off initially.



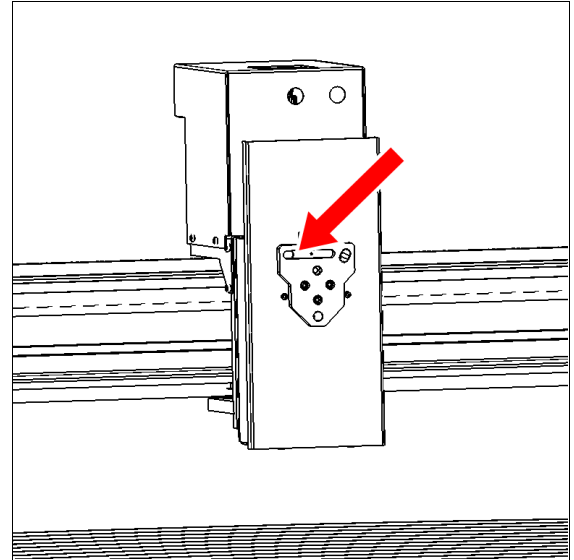
3. Touch the ground clamp lead from the plasma source to the torch cap. You should see the LED switch to green.



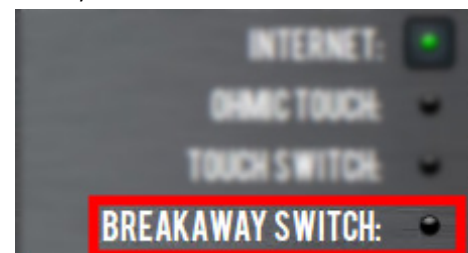
If your ohmic probe input is not responding the way described above, check the "Ohmic Probing" (page 168)

4.13.7 Testing Torch Breakaway

Your Tormach 1300PL is equipped with torch breakaway detection to stop motion in the event that a tip-up during cutting causes a collision with the torch. A proximity sensor is located in the touch mounting base to detect when the mounting clamp has been knocked off.



1. Make sure that the machine is on and PathPilot is running.
2. Switch to the Status tab on your PathPilot controller and locate the **BREAKAWAY SWITCH** led. It should be off initially.



3. Pull the torch clamp away from the base. You should see the LED light up when the torch clamp is removed and extinguish when it is replaced.

4.13.8 Power off the Machine

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

4: INSTALLATION

4.13 Verify the Installation

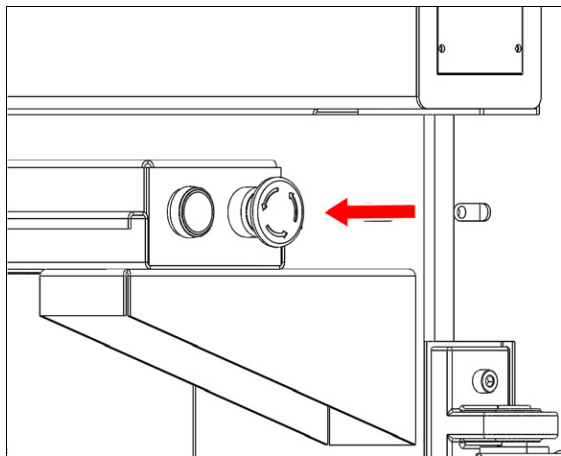


Figure 4-10: Emergency Stop button.

With the Emergency Stop button in the disabled position all motion 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**.

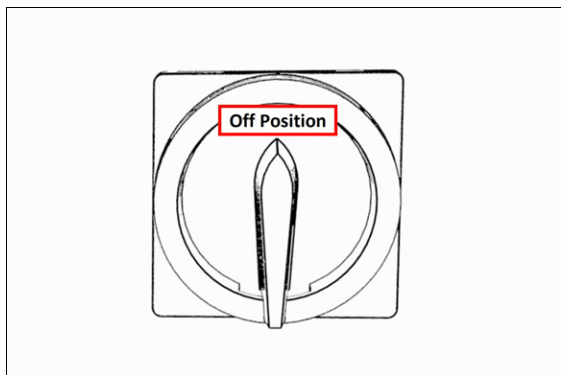


Figure 4-11: Example of the Main Disconnect switch in the Off position.

Mains power is disconnected from the machine.

4.14 SET TORCH HOME POSITION (G30)

The plasma torch will move up in Z to its home position (G30) between each cut, when the M205 command is executed. It is important to set the G30 position before running the machine, to ensure that the torch has a known position to return to.

1. Bring the machine out of E-Stop and reference all axes.
2. Move the torch to a safe Z position, higher than the slats of the water table.
3. Click the Offsets tab in PathPilot.
4. Click the "Set G30" button to save the current torch position.

In the future, when you execute an M205 or G30 command in G-Code, the torch will return to this height.

4: INSTALLATION

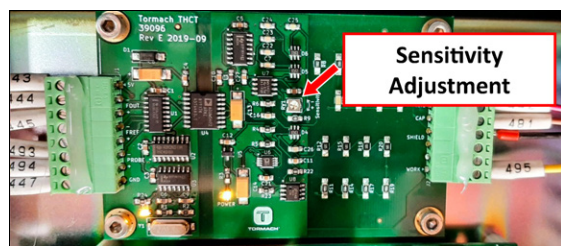
4.15 Calibrate Initial Height Sensing

4.15 CALIBRATE INITIAL HEIGHT SENSING

4.15.1 Set Ohmic Probing Sensitivity

The ohmic probing system on the 1300PL relies on a calibrated resistance value between the torch cap and the workpiece. If you do not calibrate your ohmic probe, you may find that it is triggered by water on the work piece or inside the torch cap.

1. Open the electrical cabinet and identify the Tormach THCT Board (PN 39096).

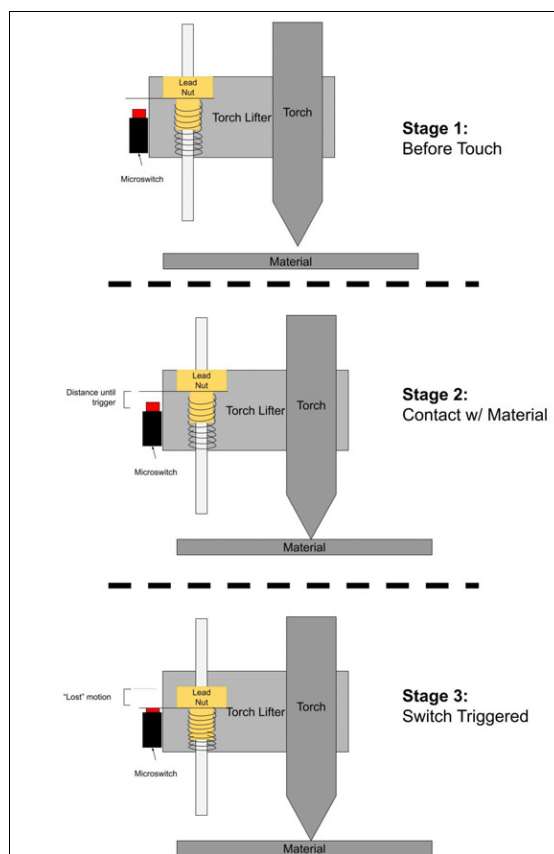


2. Select a piece of sheet metal to use as a test workpiece. Connect it to the ground clamp.
3. Jog the torch downwards in Z until the torch cap contacts the workpiece. Watch LED D8 on the bottom left of the board. If the LED lights up when the workpiece contacts the torch cap, the THCT board is measuring contact between the cap and ground as intended.
4. If the LED did not light up, identify adjustment screw RV1 on the THCT board (indicated by the arrow in the image above). Increase the sensitivity by rotating the screw clockwise, 1/16th turn at a time until the workpiece is detected.
5. Splash a small puddle of water onto your workpiece. Jog the torch cap down until it is contacting the water **but not** the workpiece itself. If the LED is on, your THCT sensitivity is too high. Decrease the sensitivity 1/16th of a turn at a time until the LED goes out.

If you have trouble adjusting the ohmic probing system or the LED never comes on, go to "Ohmic Probing" (page 168) in the Troubleshooting section.

4.15.2 Set Physical Touch Switch Trigger Depth

The torch lifter head on the 1300PL contains a spring-loaded switch to serve as a backup for ohmic probing or to be used when ohmic probing is inconvenient. The spring compresses a certain distance in Z before triggering. This distance must be measured and input on the **Settings** tab in PathPilot.



Use the procedure in "Plasma Settings Reference" (page 122) to measure and set the trigger depth.

4.16 SET UP THE PATHPILOT CONTROLLER

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

4.16.1 Specify the Date and Time

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

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



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

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

4.16.2 Specify the Keyboard Language

By default, the keyboard language is set to English.

To specify a different keyboard language:

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

4.16.3 Configure the Optional Touch Screen Kit

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

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

4.16.4 Update PathPilot

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

1. Confirm that the PathPilot controller is powered on and out of **Reset** mode.
2. Downloading and installing an update file requires an

Internet connection. From the **Status** tab, confirm that the **Internet** button LED light is on. (To configure the network, select the LED light.) Then, select **Update**.



Figure 4-12: Update button on the Status tab.

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

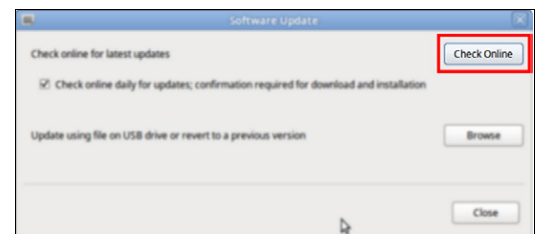


Figure 4-13: Software Update dialog box.

4. Select **Install**.



Figure 4-14: 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.

TORMACH



SYSTEM BASICS

IN THIS SECTION, YOU'LL LEARN:

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



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

CONTENTS

5.1 System Reference	66
5.2 Basic Controls Reference	68
5.3 Connectors Reference	69
5.4 Plasma Source Control	70



TORMACH®

5: SYSTEM BASICS

5.1 System Reference

5.1 SYSTEM REFERENCE

5.1.1 Water Table

The machine table is 51.2" x 51.2" (1300 mm x 1300 mm). It can be moved backwards in the frame by up to 10" (250 mm) to provide an open area for cutting large or bulky objects that do not fit on top of the water table (items like large pipes or weldments).

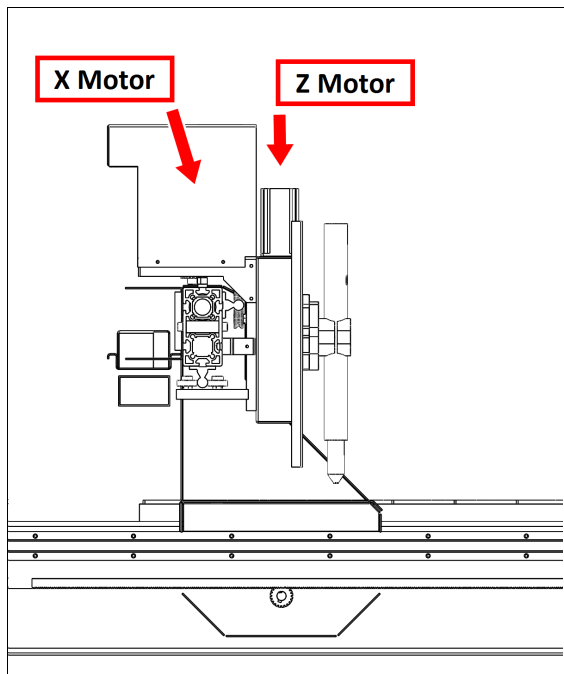


Note: Make sure you do not forget to connect the ground clamp to the workpiece when cutting something that is too large for the water table.

The water table has an internal depth of about 5.2" (132 mm) and a maximum water capacity of 59 US gallons (223 liters). Don't put more than 750 lb (340 kg) on the machine table (excluding water). Always verify that the workpiece and work-holding devices are centered on the machine table. Always lock the water table in the forward or backwards position before running a program.

5.1.2 Z Torch Head

The Z Torch head is responsible for control of torch height during cutting. It also contains the magnetic torch breakaway head and the proximity sensor required to detect a breakaway. The Z head can travel 3.9" (100 mm) vertically, which allows for cutting bulky items or even changing the height of the slats in the water table if cutting conditions require it.



The Z head is also responsible for material touch-off at the start of a cut. Touch-off on the material can occur one of two ways:

- Ohmic Touchoff
- Physical Touch Switch

Ohmic Touchoff

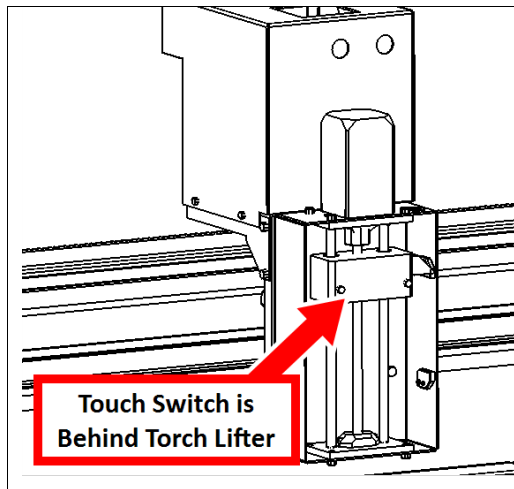
Ohmic Touchoff is a sensitive and precise method of Z positioning for initial cut height. The cap on the end of the torch is made of a conductive material (usually copper) that is linked to the machine's control system through a removable lead.

When the head touches the workpiece during the initial probing move before a cut, continuity between the torch cap and ground (through the plasma ground clamp) is detected by the control system.

This system works well for thin workpieces like 20 ga. (1 mm) and thinner sheet metal, where the weight of the torch during probing could otherwise deform the sheet metal before the probe switch triggers.

Physical Touch Switch

The physical touch switch acts as a backup to the ohmic touch system. It consists of a micro-switch inside the lifter head that detects when the spring-loaded lead nut on the Z lifter is pushed back by contact with the workpiece.



The physical touch switch will automatically be used when the ohmic touch switch does not trigger first. This can be useful in situations like cutting painted metal, where no continuity with the workpiece is possible.



Note: Even though the physical touch switch allows you to touch off on painted metal, you will still need to grind away enough paint for the ground clamp to make contact!

5: SYSTEM BASICS

5.2 Basic Controls Reference

5.2 BASIC CONTROLS REFERENCE

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

5.2.1 Machine Controls

The following controls energize the machine's control electronics:

- The Main Disconnect switch, located on the 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 operator box — which contains the blue Reset button and the red Emergency Stop button — located on the front of the machine.
When pushed in, the Emergency Stop button interrupts power to the 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 axis motion. The Reset button's LED turns on when the machine is enabled and the axis drives receive power.

5.2.2 PathPilot Interface

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

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

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

5.3 CONNECTORS REFERENCE

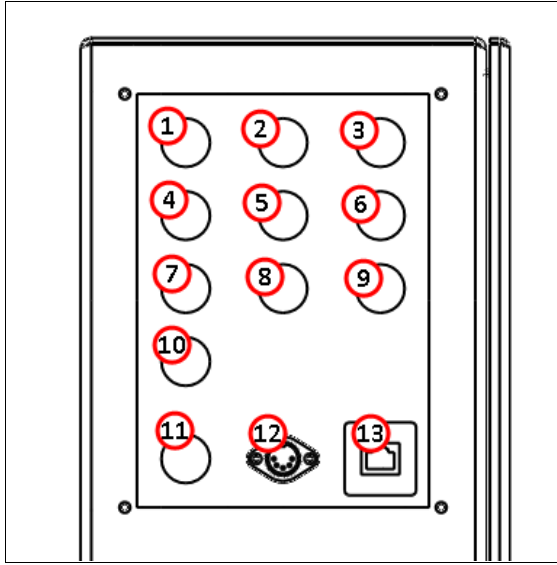


Figure 5-1: Connector panel on the upper rear of the electrical cabinet.

1. **X-Axis Motor Power Connector**
Two pin power connector for the X-axis servo.
2. **Y-Axis Motor Power Connector**
Two pin power connector for the Y-axis servo.
3. **Ohmic Cap Connector**
Input for the ohmic probing circuit. Also connects machine grounds to the cabinet ground.
4. **X-Axis Motor Control Connector**
Eight pin control connector for the X-axis servo.
5. **Y-Axis Motor Control Connector**
Eight pin control connector for the Y-axis servo.
6. **Limit Switch Inputs**
Limit switch inputs from the Z Touch Lifter. Also contains the input for the torch breakaway and touch switches.
7. **Z-Axis Motor Connector**
The Z-Axis stepper motor output.
8. **A-Axis Motor Connector**
The A-axis motor connector is used to connect to a rotary 4th axis (used for indexing or continuous 4th axis machining).
9. **24v Accessory Output**
24v outputs used to control things like pneumatic solenoids for air engravers or lifters.
10. **Emergency Stop Input**
Input cable from the operator panel containing the reset button and emergency stop.

11. **Plasma Source Control**
Control signals for the plasma source. Outputs an arc on-off signal. Torch voltage and arc-ok feedback are received here. Reference "Plasma Source Control" (on the next page)
12. **Accessory Input**
The accessory input is used to connect accessories (like probes, tool setters, and tool touch plates) to the machine.
13. **Controller Communications Port**
The controller communications port is used to connect the PathPilot controller to the machine. The controller communications port (and the cable that connects to it) sends all communication between the PathPilot interface and the machine.

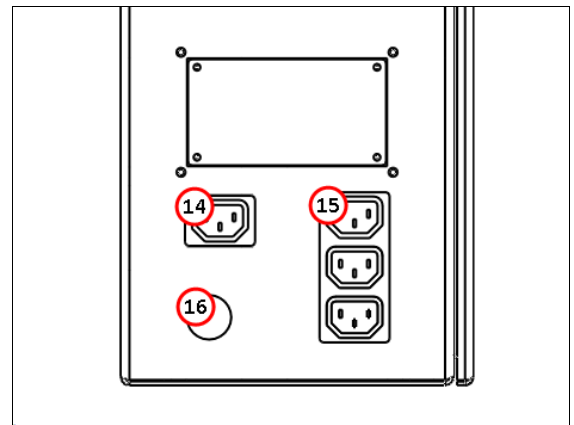


Figure 5-2: Connector panel on the lower rear of the electrical cabinet.

14. **Switched Power Port**
The IEC-320 switched power port is used to supply power to accessories that can be controlled by PathPilot such as a downdraft extractor fan. This outlet outputs the same voltage as the machine input voltage (230 VAC).
15. **Accessory Power Port**
The IEC-320 accessory power ports are used to supply power to peripheral accessories (like the PathPilot controller and monitor). These outlets output the same voltage as the machine input voltage (230 VAC).
16. **Electrical Power Input**
Input power lead for the electrical cabinet.

5: SYSTEM BASICS

5.4 Plasma Source Control

5.4 PLASMA SOURCE CONTROL

The 1300PL uses a 7-pin connector for controlling the plasma source. A 7-pin to 14-pin cable is included with your machine to control Hypertherm plasma sources.

The pin-out of the 7-pin connector on the electrical cabinet is as follows:

Pin	Type	Signal	Notes
1	Output	Start Plasma	Dry contact closure. Closed relay signals plasma on.
2			
3	Input	Arc Voltage -	An internal 50:1 voltage divider in the plasma source is required .
4		Arc Voltage +	
5	Input	Arc-Ok	Expects dry contact closure at plasma source. Closed contacts signal arc-ok.
6			
7	Not Used		

The pin-out of the 14-pin Hypertherm connector on the end of the cable is as follows:

Pin	Type	Signal	Notes
3	Output	Start Plasma	Dry contact closure. Closed relay signals plasma on.
4			
5	Input	Arc Voltage -	An internal 50:1 voltage divider in the plasma source is required .
6		Arc Voltage +	
12	Input	Arc-Ok	Expects dry contact closure at plasma source. Closed contacts signal arc-ok.
14			

PATHPILOT INTERFACE OVERVIEW

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

6.1 About PathPilot	72
6.2 Notebook Section	73
6.3 Persistent Controls	80
6.4 Keyboard Shortcuts	84
6.5 Manage PathPilot Versions	85

6: PATHPILOT INTERFACE OVERVIEW

6.1 About PathPilot

6.1 ABOUT PATHPILOT

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

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

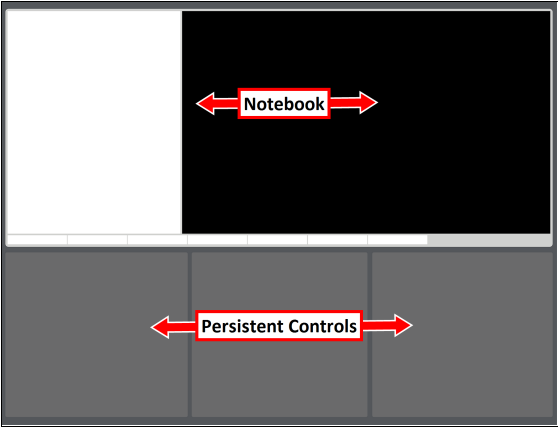


Figure 6-1: Sections in the PathPilot interface.

6.2 NOTEBOOK SECTION

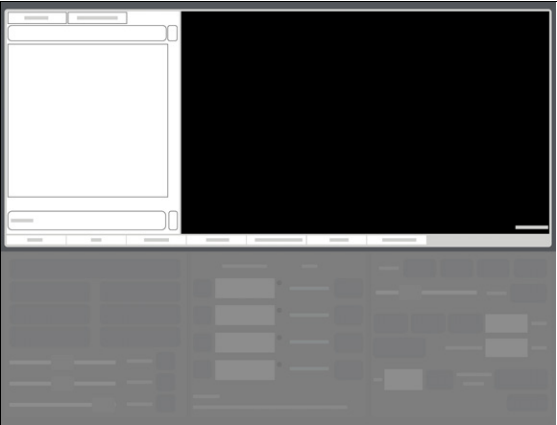


Figure 6-2: Notebook section.

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

6.2.1 Main Tab	74
6.2.2 File Tab	75
6.2.3 Settings Tab	76
6.2.4 Offsets Tab	77
6.2.5 Conversational Tab	78
6.2.6 Status Tab	79

6: PATHPILOT INTERFACE OVERVIEW

6.2 Notebook Section

6.2.1 Main Tab

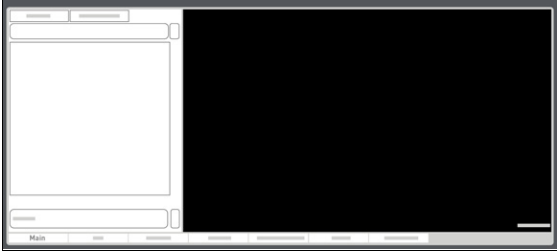


Figure 6-3: Main tab.

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

- Access G-code files that are already loaded into PathPilot, and open or close them.
For information, see "Access Recent G-Code Files" (page 89); "Close the Current Program" (page 89).
- Send G-code commands directly to the machine using the Manual Data Input (MDI) Line DRO field.
For information, see "Manually Enter Commands" (page 113).
- 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 90); "Expand the G-Code Tab" (page 89); "Change the View of the Tool Path Display" (page 91).

6.2.2 File Tab



Figure 6-4: File tab.

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

- Transfer G-code files into the PathPilot controller.
For information, see "Transfer Files to and From the Controller" (page 88).
- Edit G-code files.
For information, see "Edit G-Code with a Text Editor" (page 89).
- Load .nc files into PathPilot to run a program.
For information, see "Load G-Code" (page 88).
- Move files within the system.
For information, see "Preview G-Code Files" (page 88);
"Manage System Files" (page 117).

6: PATHPILOT INTERFACE OVERVIEW

6.2 Notebook Section

6.2.3 Settings Tab

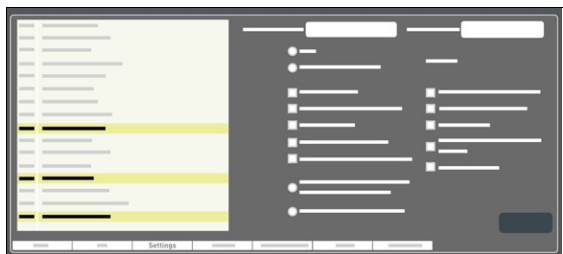


Figure 6-5: Settings tab.

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

- Change the network name with which you're using PathPilot.
For information, see "Change the Network Name" (page 97).
- Change the screen's layout orientation (landscape or portrait).
For information, see "Change the Screen Orientation" (page 97).
- Configure PathPilot for the accessories you're using.
For information, see "Enable the On-Screen Keyboard" (page 100); "Enable the USB M-Code I/O Interface Kit" (page 101); "Use a USB Camera" (page 101).
- Specify the way in which you want to use a G30 move.
For information, see "Limit G30 Moves" (page 100).
- Identify the available G-code modes that you can use.
For information, see "View Available G-Code Modes" (page 106).
- Adjust plasma settings.
For information, see "Set Torch Touch Trigger Depth" (page 98); "Set Minimum Touchoff Spacing" (page 99); "Enable Ohmic Touchoff" (page 99); "Enable Torch Height Control" (page 99); "Enable Arc-Ok Checking" (page 99).

6.2.4 Offsets Tab

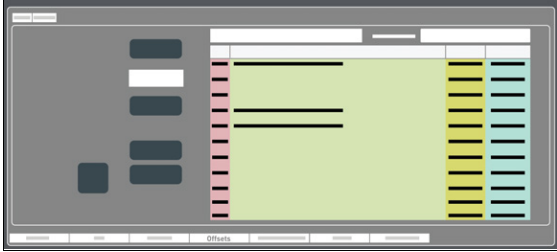


Figure 6-6: Offsets tab.

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

- Make and restore backup files of your settings.
For information, see "Create Backup Files" (page 117);
"Restore Backup Files" (page 118).
- Import and export .csv files of your tool table.
For information, see "Import and Export the Tool Table" (page 119).
- Preset a G30 position.
For information, see "Use a G30 Position" (page 113).
- Read the currently programmed work offsets.
For information, see "View Work Offsets" (page 105).

6: PATHPILOT INTERFACE OVERVIEW

6.2 Notebook Section

6.2.5 Conversational Tab



Figure 6-7: Conversational tab.

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

- Import a .dxf file.
For information, see "Import a DXF File" (page 92).

6.2.6 Status Tab



Figure 6-8: 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 96).
- Update or install a previous version of PathPilot.
For information, see "Manage PathPilot Versions" (page 85).

6: PATHPILOT INTERFACE OVERVIEW

6.3 Persistent Controls

6.3 PERSISTENT CONTROLS



Figure 6-9: 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:

6.3.1 Program Control Area.....	81
6.3.2 Position Status Area.....	82
6.3.3 Manual Control Area.....	83

6.3.1 Program Control Area



Figure 6-10: 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 107).
- Start, stop, or pause a G-code program.
For information, see "Start a Program" (page 109); "Stop Machine Motion" (page 109); "Use the Feed Hold Function" (page 111).
- Use overrides to change the feed rate, voltage, and maximum velocity.
For information, see "Use the Feed Rate Override Function" (page 111); "Use the Maxvel Override Function" (page 112); "Use the Voltage Override Function" (page 112).
- Manually control a G-code program.
For information, see "Use M01 Break Mode" (page 112); "Use Single Block Mode" (page 112).

6: PATHPILOT INTERFACE OVERVIEW

6.3 Persistent Controls

6.3.2 Position Status Area



Figure 6-11: Position Status area.

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

- Reference the machine axes.
For information, see "Reference the Machine" (page 108).
- Create work offsets.
For information, see "Set Work Offsets" (page 105).
- Understand how you're jogging the machine.
For information, see "View the Active Axis to Jog" (page 107); "View the Current Machine Position" (page 108); "View the Distance to Go" (page 110).
- Quickly determine which G-code modes are active.
For information, see "View the Active G-Code Modes" (page 109).

6.3.3 Manual Control Area



Figure 6-12: 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 107).
- Use automatic look-up tables to find feed rate, amperage, pierce parameters, and torch height control voltage for the material you're cutting.
For information, see "Use the AutoFS Material Picker" (page 128).

6: PATHPILOT INTERFACE OVERVIEW

6.4 Keyboard Shortcuts

6.4 KEYBOARD SHORTCUTS

The following table lists the keyboard shortcuts in PathPilot.

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

6.5 MANAGE PATHPILOT VERSIONS

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

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

6.5.1 Download and Install an Update File from the Controller

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

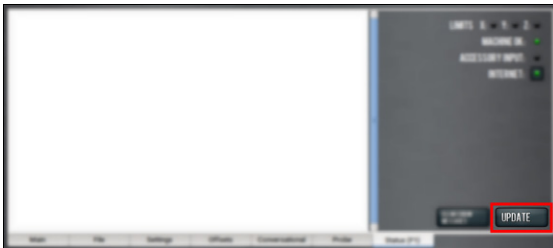


Figure 6-13: Update button on the Status tab.

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

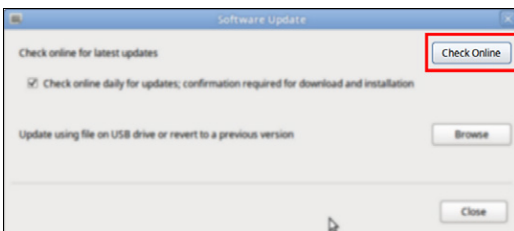


Figure 6-14: Software Update dialog box.

4. Select **Install**.

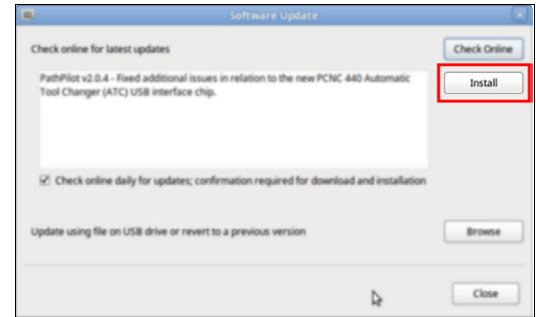


Figure 6-15: Install button on the Software Update dialog box.

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.

6.5.2 Install an Update File from a USB Drive

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

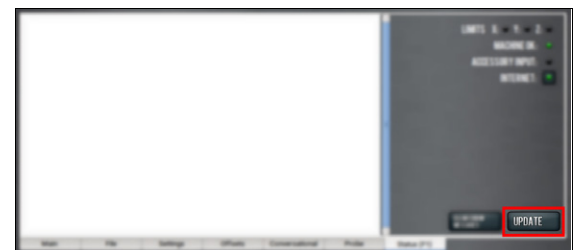


Figure 6-16: Update button on the Status tab.

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

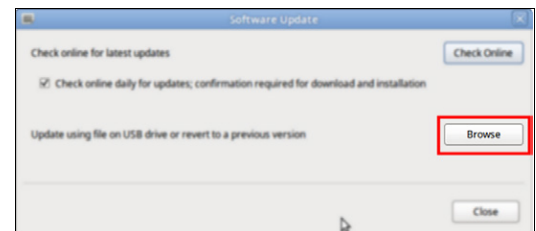


Figure 6-17: Software Update dialog box.

6: PATHPILOT INTERFACE OVERVIEW

6.5 Manage PathPilot Versions

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



Figure 6-18: 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.

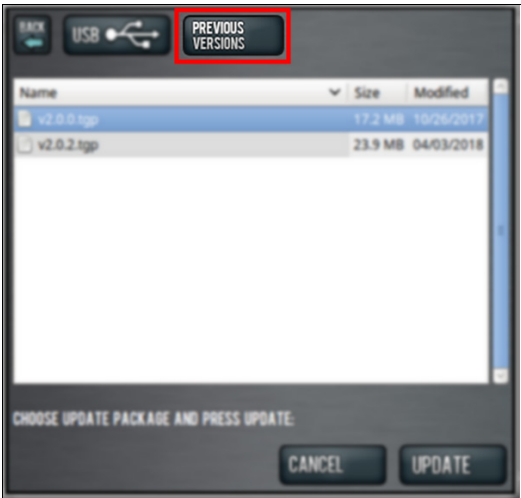


Figure 6-21: Browse dialog box.

- 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.

6.5.3 Install a Previous Version of an Update File

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

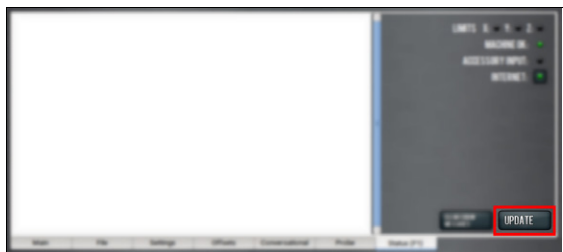


Figure 6-19: Update button on the Status tab.

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

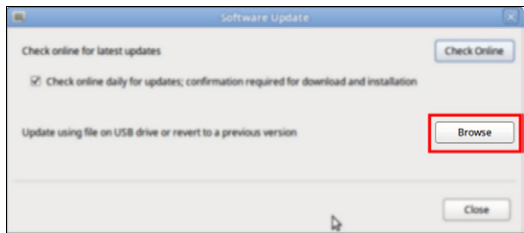


Figure 6-20: Software Update dialog box.

- 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

7.1 Create and Load G-Code Files	88
7.2 Machine Settings and Accessories	96
7.3 Set Up G-Code Programs	105
7.4 Run G-Code Programs	107
7.5 Control G-Code Programs	111
7.6 System File Management	117

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

7.1 CREATE AND LOAD G-CODE FILES

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

7.1.1 Load G-Code	88
7.1.2 Edit G-Code with a Text Editor	89
7.1.3 Read G-Code	89
7.1.4 Import a DXF File	92

7.1.1 Load G-Code

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

To load G-code:

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

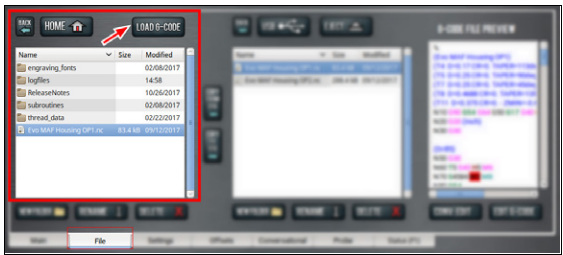


Figure 7-1: Controller Files window on the File tab.



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

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

Transfer Files to and From the Controller

To run a G-code program, you must transfer the files to the PathPilot controller. 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).

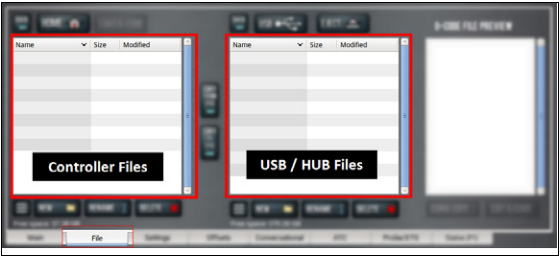


Figure 7-2: File tab.



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 →**.

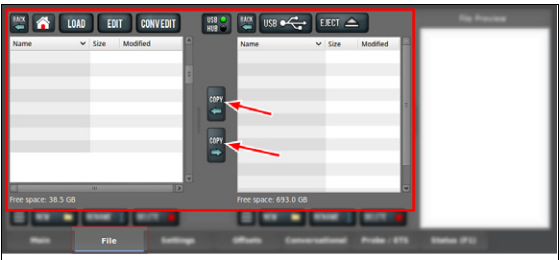


Figure 7-3: File tab.



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.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

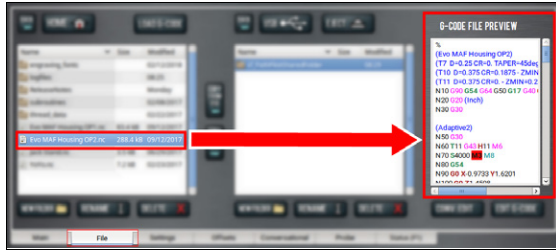


Figure 7-4: File tab.

Access Recent G-Code Files

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

To access recent G-code files:

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

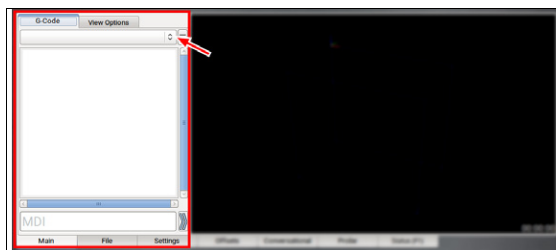


Figure 7-5: Recent Files menu on the Main tab.

The last five program files loaded into PathPilot display.

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

Close the Current Program

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



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

The currently loaded G-code program closes.

7.1.2 Edit G-Code with a Text Editor

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

To edit G-code with a text editor:

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

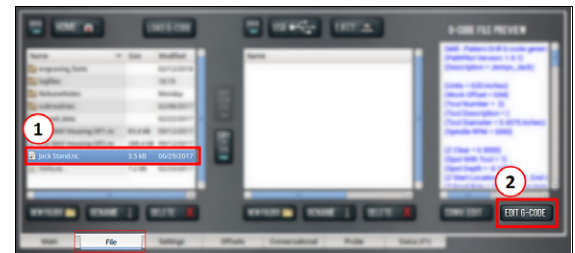


Figure 7-7: Edit G-code button on the File tab.

The file opens in a text editor.

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

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

7.1.3 Read G-Code

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

Expand the G-Code Tab	89
Search in the Code	90
Set a New Start Line	91
Change the View of the Tool Path Display	91

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" (on the next page).

To expand the G-code tab:

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

- Select the **Window Expander**.

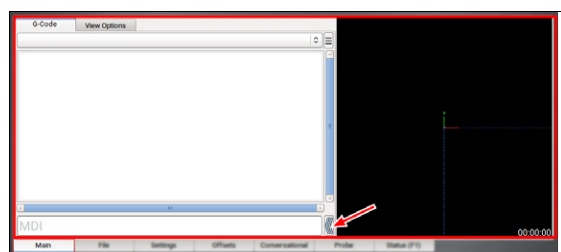


Figure 7-8: Window Expander on the Main tab.

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" (on the next page).
- **Orange Line** Indicates the line of code that PathPilot is currently executing.

Search in the Code

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

To search in the code:

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

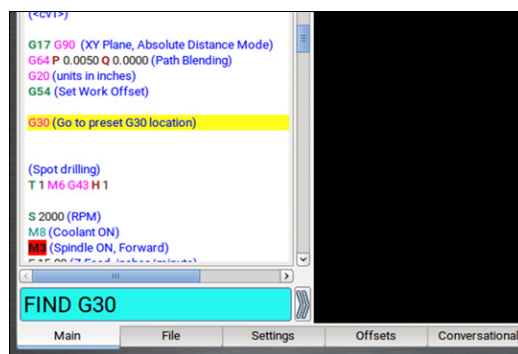


Figure 7-9: Search for a text command.

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

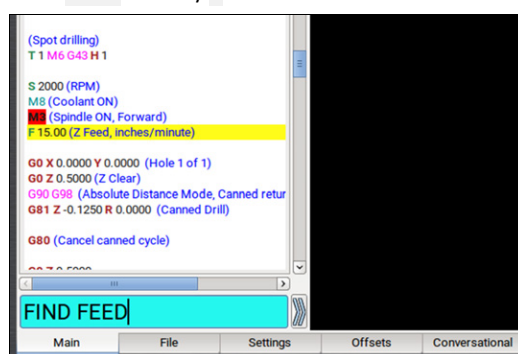


Figure 7-10: Search for a feed command.

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



Note: The find command is not case-sensitive.

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



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

Set a New Start Line

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

To set a new start line:

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

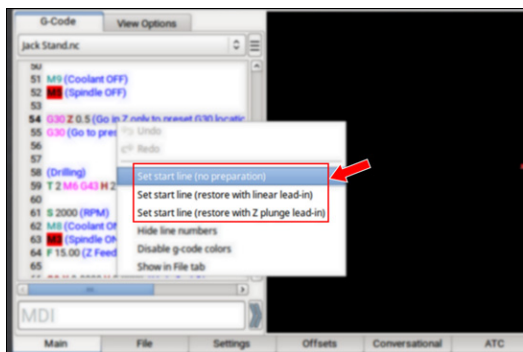


Figure 7-11: Accessing the Options menu by right-clicking.

- 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 and Y, then Z to the current position, then feeds in a straight linear line to the start line position.



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



EXAMPLE

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

- **Set start line (restore with Z plunge lead-in)**

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



EXAMPLE

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

Change the View of the Tool Path Display

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

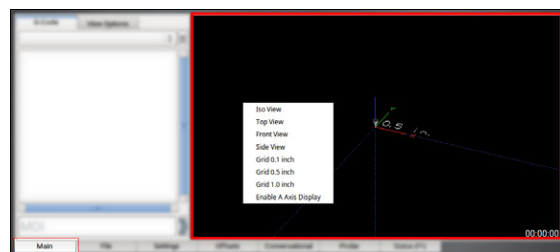


Figure 7-12: Tool Path display on the Main tab.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

- Select the **View Options** tab.

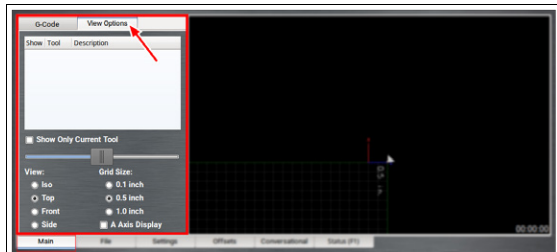


Figure 7-13: View Options tab on the Main tab.

2. Select a new view.

For information, see "About the Tool Path Display" (below).

About the Tool Path Display

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

Depending on which programming mode you're in (G20 or G21), PathPilot defaults to one of the following grid line spacings:

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

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

- **Dotted Blue Lines** Indicate the boundary box (the ends of travel of the axes).
- **Red Lines** Indicate the tool path as it is cut.



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

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

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

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

7.1.4 Import a DXF File

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

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



Figure 7-14: DXF tab on the Conversational tab.

2. Select the **File** DRO field.

The **File Selector** dialog box opens.

3. Select the .dxf file, and then select **Open**.
4. The shapes from the selected file are loaded into the **Preview** window.



Note: The .dxf file must already be transferred to the PathPilot controller. For information, see "Transfer Files to and From the Controller" (page 88).

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



EXAMPLE

If you type 1.0 in the **Scale** DRO field, the .dxf is scaled at 100%.

If you type 2.0 in the **Scale** DRO field, the .dxf is scaled at 200%.

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

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

8. Select one of the following to set the cutter compensation to be applied to the tool path:
 - **On:** The tool moves along the path.
 - **Outside / Right:** Offsets the tool path right of the drawing path, seen from the direction where the tool enters the path.
 - **Inside / Left:** The opposite of **Outside / Right**.

Working with Layers and Shapes

The .dxf file contains shapes grouped into layers.

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

Change the Layer or Shape Cut Order

- Use the **Up Arrow** and **Down Arrow** buttons above the **Shape Selection** tree view window. Shapes or layers higher in the tree view window are cut earlier than those below it. The order in which the shapes are cut is the same as the order of the enabled shapes in the tree view window and the cyan path in the **Preview** window.



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

Adjust the Tree View Window

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

Working in the Preview Window

The **Preview** window uses the following colors:

- **Cyan** Selected paths
- **Gray** Disabled paths
- **White** Drawing path
- **Magenta** Cut path
- **Dark Cyan Stippled Line** Tool path between cuts
- The coordinates use the following colors:
 - **Red** X-axis
 - **Green** Y-axis
 - **Blue** Z-axis

Create and Add Shape Library Templates

Starting with PathPilot v2.9.0, you can create and add new templates to the Shape Library system.

Templates consist of two files:

- **.template:** A parametric G-code file that serves as the actual template.
- **.png:** A thumbnail image to identify it in the shape library.



Tip! We recommend that you create templates externally in your favorite G-code editor and then copy them to your PathPilot controller like you would any other file.

Create a Template

1. Create a new file named **[shape_name].template**, where **[shape_name]** is your desired template name.



EXAMPLE

To create a template for a welding bracket, make the file name **welding_bracket.template**.

7: PATHPILOT TOOLS AND FEATURES

7.1 Create and Load G-Code Files

2. Inside the file that you created in Step 1, you can write regular G-code with the addition of special parametric variables that PathPilot uses to display adjustable parameters in the Shape Library.
- The following example is a template file that creates a line from one (X, Y) location to another:

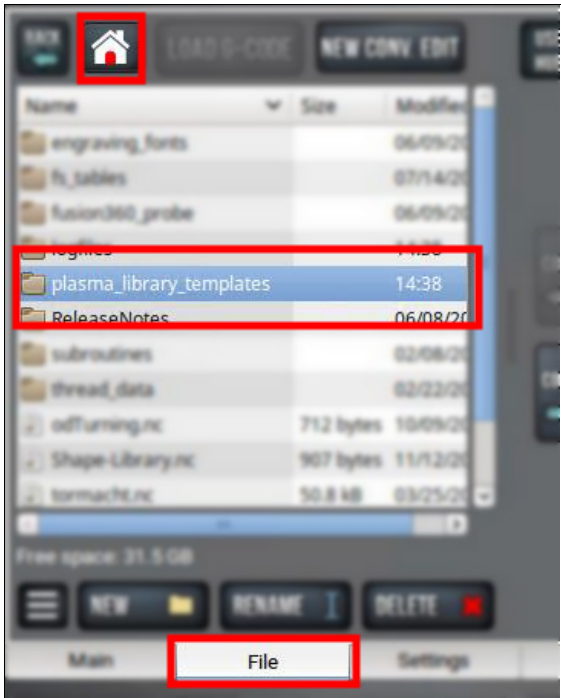
```
line.templateText#<start_x> = 1.0
(PARAMFLOAT)
#<start_y> = 1.0 (PARAMFLOAT)
#<end_x> = 5.0 (PARAMFLOAT)
#<end_y> = 5.0 (PARAMFLOAT)
G20 (set units to inches)
G91.1 (Restore Incremental Arc Distance Mode)
G90 (Absolute Distance Mode)
G30
M200
G0 X[#<start_x>] Y[#<start_y>]
G15
G16
(Cut to end of line)
G1 X[#<end_x>] Y[#<end_y>]
M205
```

Note: The special lines at the top each create a parameter that PathPilot recognizes and displays on screen. Each parameter can either have the type **PARAMFLOAT** or **PARAMINT** for floating point numbers or integers, respectively.

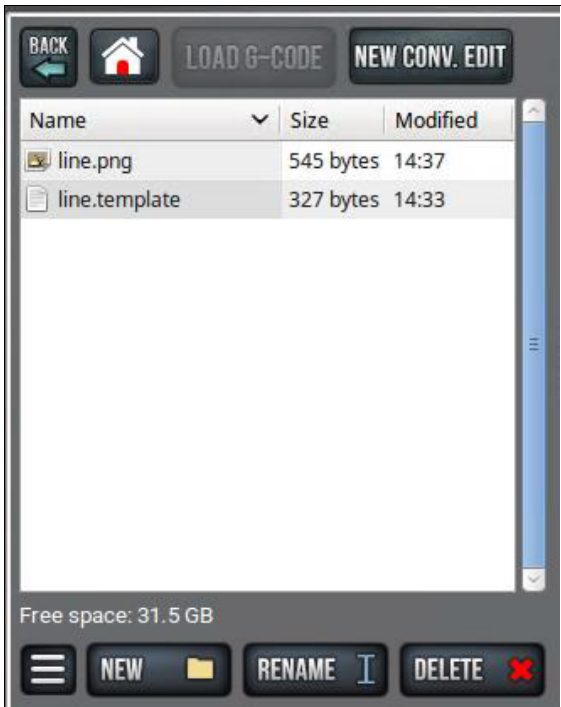
3. (Optional) Once you've written the G-code for your template, you can create a **.png** image to use as the thumbnail. The image must:
- Be sized to 256 pixels × 256 pixels.
 - Use a file name that matches the template name.

EXAMPLE
A thumbnail for the template **welding_bracket.template** requires an image with the file name **welding_bracket.png**.

you've updated to the latest version of PathPilot.



2. Copy the **.template** file and the **.png** file that you created into the **plasma_library_templates** folder.

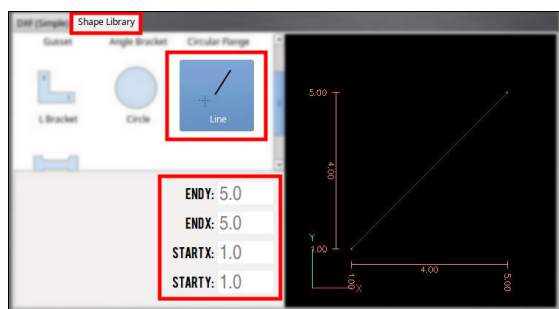


Add a Template

1. From the PathPilot controller, on the **File** tab, find the **plasma_library_templates** folder inside the **Home** directory. If you don't see this folder, make sure that

3. Restart PathPilot.

Your new template displays on the **Shape Library** tab, with any defined parameters from the **.template** file displayed as configurable options.



7: PATHPILOT TOOLS AND FEATURES
7.2 Machine Settings and Accessories

7.2 MACHINE SETTINGS AND ACCESSORIES

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

7.2.1 Enable an Internet Connection 96
7.2.2 Enable Automatic Updates.....96
7.2.3 Change the Network Name 97
7.2.4 Change the Screen Orientation 97
7.2.5 Set Torch Touch Trigger Depth 98
7.2.6 Set Minimum Touchoff Spacing 99
7.2.7 Enable Ohmic Touchoff..... 99
7.2.8 Enable Torch Height Control..... 99
7.2.9 Enable Arc-Ok Checking 99
7.2.10 Disable Hard Stop Referencing 100
7.2.11 Limit G30 Moves..... 100
7.2.12 Enable the On-Screen Keyboard 100
7.2.13 Enable the USB M-Code I/O Interface Kit 101
7.2.14 Enable Tooltips..... 101
7.2.15 Use a USB Camera..... 101

7.2.1 Enable an Internet Connection

If desired, you can enable an internet connection on your PathPilot controller. An internet connection allows you to receive automatic PathPilot updates and transfer files with PathPilot HUB instead of a USB drive. To enable an internet connection:

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



Figure 7-15: Internet button on the Settings tab. The Network Configuration dialog box displays.

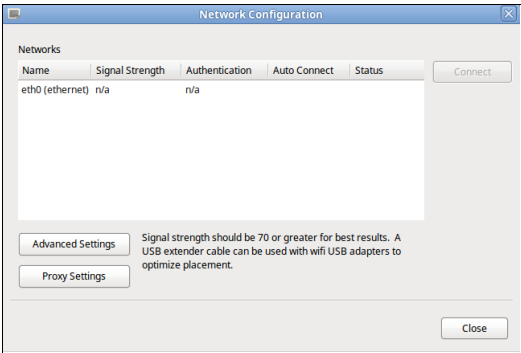


Figure 7-16: Network Configuration dialog box.

- 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.

7.2.2 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.

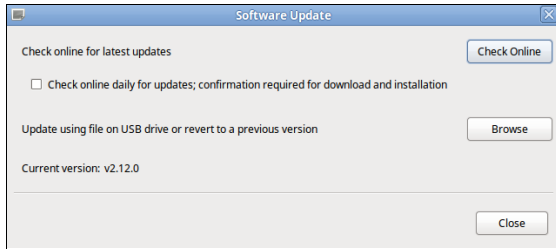


Figure 7-17: Software Update dialog box.

- From the **Software Update** dialog box, select the **Check online daily for updates; confirmation required for download and installation** checkbox.
- Select **Close**.
When future updates are available, the **Status** tab displays a notification.

7.2.3 Change the Network Name

If you're connected to a network using either the Ethernet jack or the (optional) Wireless Network Adapter (PN 38207), the PathPilot controller appears on your network as **network-attached storage**. The default network name of the controller is **TORMACHPCNC**.

To change the network name:

- From the **Network Name** field, type a new network name.

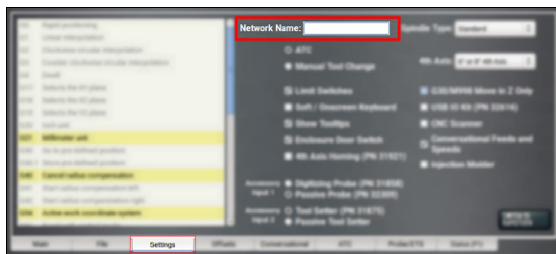


Figure 7-18: Network Name field on the Settings tab.



Note: The network name must be unique within your network.

- Select the **Enter** key.
- For the change to take effect, you must restart the controller.

7.2.4 Change the Screen Orientation

A vertical orientation for 1920 × 1080 monitors is supported in PathPilot v2.10.0 and later. For more information on the portrait layout, go to "About Portrait Screen Layout" (below). To change the screen orientation:

- From the PathPilot interface, on the **Settings** tab, select **Portrait** from the **Layout** drop-down menu. Restart the controller.

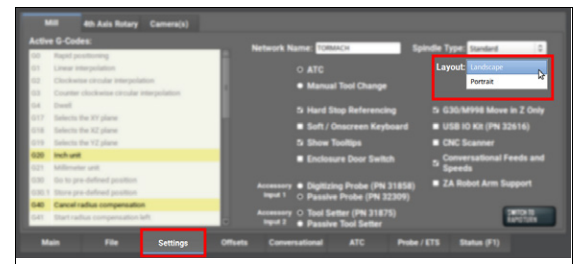


Figure 7-19: Layout drop-down menu on the Settings tab.

- Rotate the monitor to the portrait orientation. You can rotate it either left or right, depending on what's easier for your setup.
- While the controller is restarting, specify which direction you've rotated the monitor. Select **Apply**. If the result is unexpected, click **Restore Previous Configuration** on the confirmation dialog and choose a rotation direction again.

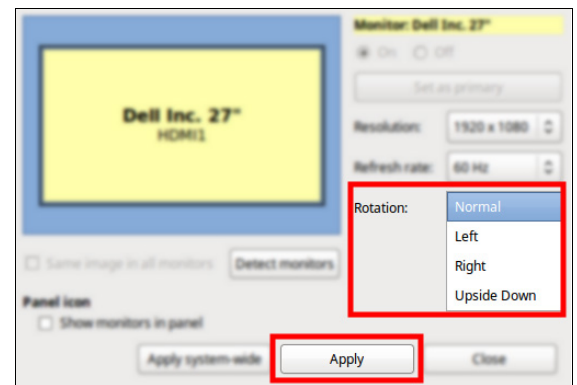


Figure 7-20: Monitor configuration dialog box.

The controller restarts in portrait layout.

About Portrait Screen Layout

Portrait layout provides some key advantages:

- A larger tool path window that's always visible at the top of the screen, regardless of which tab you have active.

7: PATHPILOT TOOLS AND FEATURES
7.2 Machine Settings and Accessories

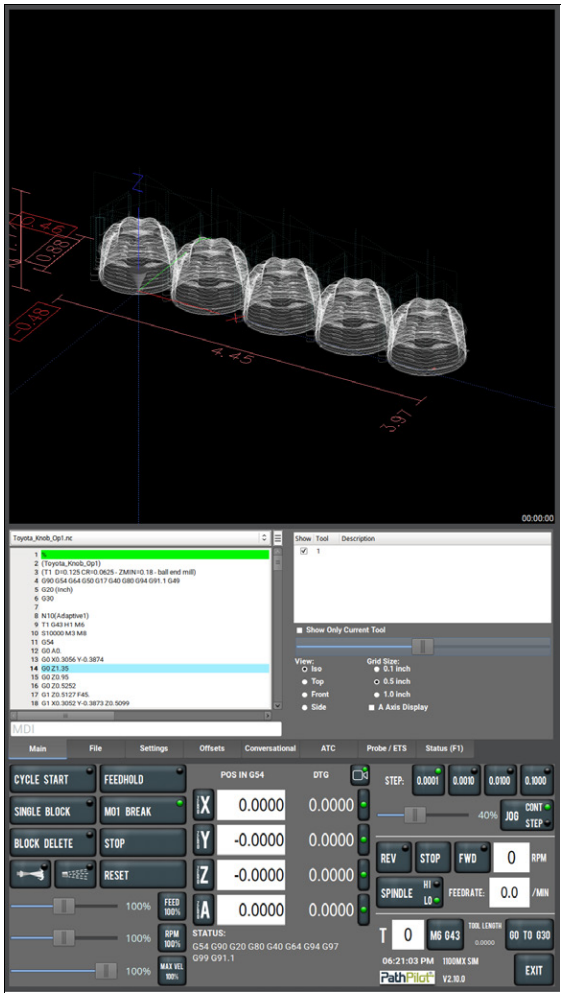


Figure 7-21: Tool Path window in portrait screen layout.

- A wider G-code window to more easily read the loaded G-code file and, if enabled, line numbers.
- The tool path window's view options are always visible for much easier access.
- When browsing G-code files using the File tab, file previews display on the top portion of the screen.

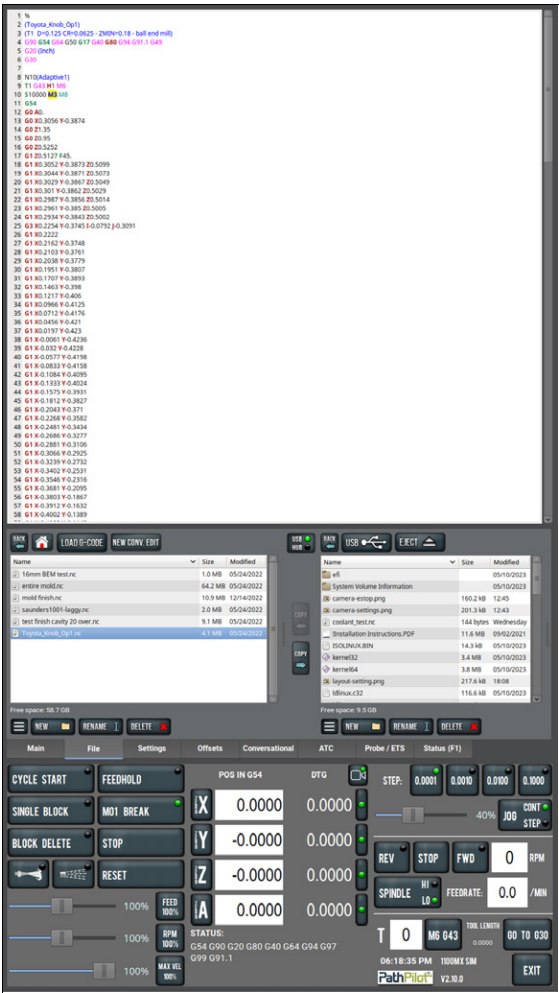
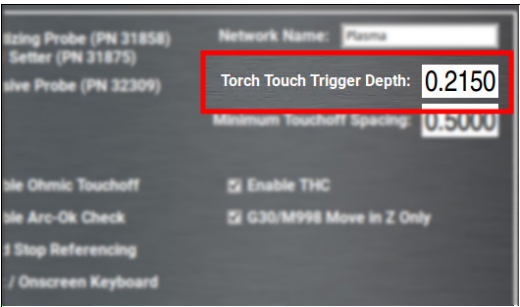


Figure 7-22: File tab G-code preview in portrait screen layout.

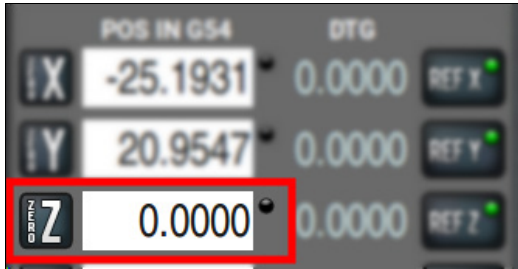
7.2.5 Set Torch Touch Trigger Depth



When using the physical touchoff switch for material height sensing, there's a certain amount of lost motion in the Z-axis between when the torch head touches the material and when the micro-switch in the torch lifter triggers. It's important to set an accurate value for the distance between torch touch and touch switch triggering so that the initial pierce height can be correctly computed.

To set torch touch trigger depth:

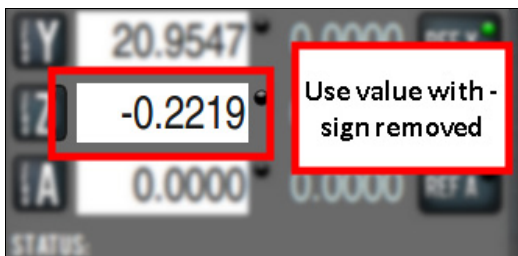
1. Power on the machine and reference all axes.
2. From the PathPilot interface, on the **Settings** tab, disable **Ohmic Touchoff**.
3. Put a piece of material on the machine table and position the torch above it.
4. Using a slow jog speed (~5 IPM), jog the torch down until it's just contacting the material.
5. Zero your Z-axis at this position.



6. Switch to the **Status** tab and watch the **Touch Switch** LED. Very slowly, jog downwards until the LED comes on.

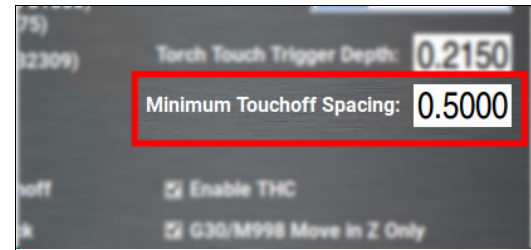


7. Note the value in the **Z DRO** field. This is your Torch Touch Trigger Depth. Convert this value to a positive number and enter it on the **Settings** tab.



8. Re-enable **Ohmic Touchoff** on the **Settings** page.

7.2.6 Set Minimum Touchoff Spacing



Minimum touchoff spacing is a time saving setting designed to avoid unnecessary probing during programs with a large number of individual cuts.

After the machine has probed for initial cut height, if the next cut begins within this distance it will be assumed that the material height is the same. This allows the machine to skip the probing routine and proceed straight to pierce height. If your material is very flat and uniform, you can use a large distance for this setting (>2 in.). If your material is uneven or you would simply prefer to probe for height every time a cut begins, enter zero for this value.

7.2.7 Enable Ohmic Touchoff

This setting disables the ohmic sensing system entirely and relies solely on the physical touch switch for height sensing. This can be useful for cutting programs that result in water being repeatedly splashed into the torch nozzle, causing spurious probing trips.



Note: Do not disable ohmic touch-off when cutting very thin material. The weight of the torch will bend the material during probing and cause inaccurate pierce height.

7.2.8 Enable Torch Height Control

Disabling this setting stops all dynamic height adjustment during cutting. The torch will stay at the same height for the length of a cut and not adjust to follow the contours of the material.

7.2.9 Enable Arc-Ok Checking

Hypertherm plasma sources supply an arc-ok signal to PathPilot when the cutting arc is fully established. When this setting is enabled, the machine will pause during a G16 pierce until it receives an arc-ok signal.

Disable this setting if your plasma source does not supply an arc-ok signal.

7: PATHPILOT TOOLS AND FEATURES

7.2 Machine Settings and Accessories

7.2.10 Disable Hard Stop Referencing

To provide a temporary workaround for a malfunctioning limit switch circuit, you can disable the limit switches.



Note: By default, the Hard Stop Referencing checkbox is selected.

To disable :

- 1. From the **Settings** tab, clear the **Hard Stop Referencing** checkbox.

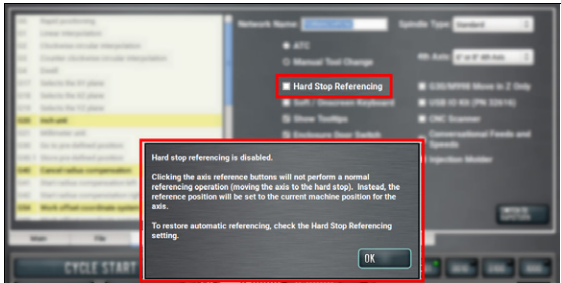


Figure 7-23: Hard Stop Referencing checkbox on the Status tab.

- 2. Select **OK**.
The machine completes a unique referencing procedure after selecting the axis reference buttons: rather than moving each axis to the end of its travel, the reference position is set as the machine's current position.



Tip! This is useful for troubleshooting, because you're now able to move the axis.

7.2.11 Limit G30 Moves

You can limit G30 moves so that only the Z-axis moves. For information, see "About G30" (page 113).

To limit G30 moves:

- From the **Settings** tab, select **G30/M998 Move in Z Only**.

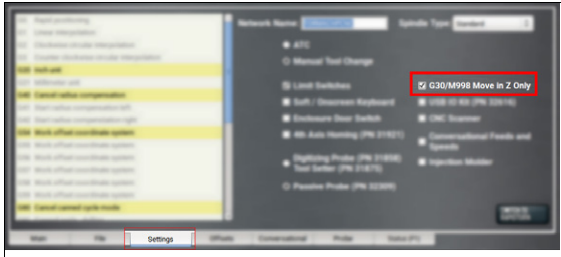


Figure 7-24: Settings tab.

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 113).
Use a G30 move to start a coordinated movement of the axes. You can limit the movement to only the Z-axis. For information, see "Limit G30 Moves" (above).



Tip! It's useful to program a G30 move right before a tool change so that the machine can jog to a safe tool change position.

7.2.12 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" (on the next page).

To enable and use the soft (on-screen) keyboard:

- 1. From the **Settings** tab, select **Soft / On-Screen Keyboard**.

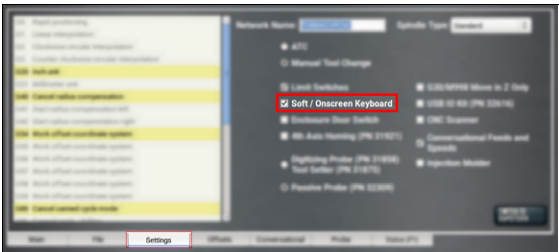
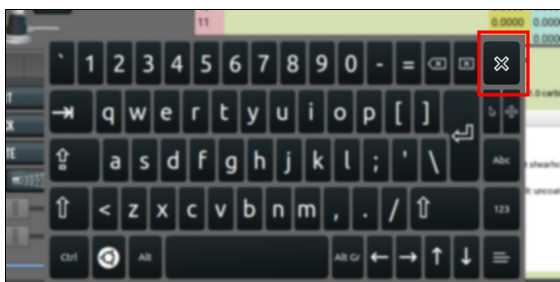


Figure 7-25: Settings tab.

- 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.



- To close the keyboard, select the **X** key.



About Soft Keyboards

If you enabled a soft keyboard (on-screen keyboard) in the PathPilot interface to use with an optional touch screen or operator console, a keyboard opens when you select any field where keyboard input is required.

The keyboard displays a wide range of keys: both uppercase and lowercase, symbols, arrow keys, caps lock, backspace and delete, and more.



Figure 7-26: Soft (on-screen) keyboard.

7.2.13 Enable the USB M-Code I/O Interface Kit

If you have a USB M-Code I/O Interface Kit (PN 32616), you must first enable it in the PathPilot interface.

To enable the USB M-Code I/O Interface Kit:

- From the **Settings** tab, select **USB IO Kit (PN 32616)**.

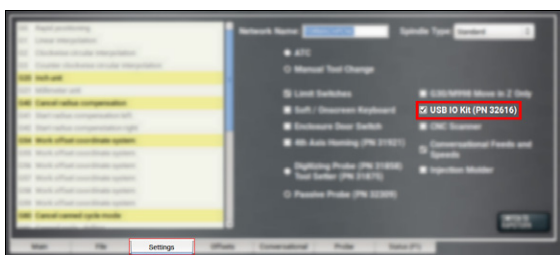


Figure 7-27: Settings tab.

7.2.14 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:

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

Figure 7-28: Show Tooltips checkbox.



Note: If you disable the tooltips, you can still display them for specific items. Hover over an area of the interface, and select the Shift key on the keyboard.

7.2.15 Use a USB Camera

After plugging in the USB camera, navigate to the camera settings. From the PathPilot interface, in the **Settings** tab, open the **Camera(s)** tab. Identify the **Camera Status** read-only dialog box.



Figure 7-29: 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.

7: PATHPILOT TOOLS AND FEATURES
7.2 Machine Settings and Accessories

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 7-30: Manual recording controls.

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



Figure 7-31: 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:

- 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 7-32: Camera settings.

To take a picture (using all of the USB cameras at once):

- Select Snapshot in the Manual Recording area of the Camera(s) tab. The Main tab displays.
- Review the camera images, which display on top of the Tool Path area. The camera images refresh every 0.5 seconds.

- Align the cameras or adjust lighting to your preference, and then select the **Shutter** button.

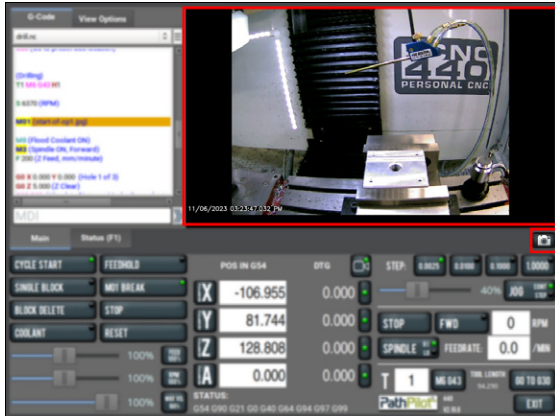


Figure 7-33: Example of taking a photo.

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:

- Toggle the **Automatic e-stop loop recording** checkbox in the **Camera Settings** area of the **Camera(s)** tab.



Note: This feature is enabled by default.

Automatic E-stop loop recording starts when the **Reset** button is selected. If you selected **Video Camera Recording** to abort a previous E-stop loop recording, select **Reset** to start it again.

To view E-stop videos:

- 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

- 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

7: PATHPILOT TOOLS AND FEATURES

7.2 Machine Settings and Accessories

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 150).

7.3 SET UP G-CODE PROGRAMS

Before running a G-code program, you must first make sure that the machine is properly set up for the specific G-code program.

7.3.1 Set Work Offsets.....	105
7.3.2 View Work Offsets.....	105
7.3.3 View Available G-Code Modes.....	106

7.3.1 Set Work Offsets

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

- Select **Zero [Axis]**.

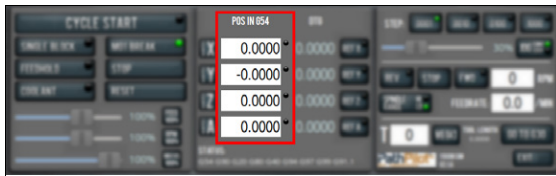


Figure 7-34: Work Offset DRO fields.

To change work offsets:

1. On the **Main** tab, in the **MDI Line** DRO field, type the new work offset to activate (for example, G55). Then select the **Enter** key.
2. The new work offset displays in the following locations in the PathPilot interface:
 - The **Status** read-only DRO field.
 - Above the **Work Offset** DRO fields.



Figure 7-35: 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).

About Work Offsets

Work offsets allow you to think in terms of X, Y, and Z coordinates with respect to the part, rather than thinking of them with respect to the machine position. This means that you can jog the machine to an arbitrary location (like the end of a workpiece) and call that location zero.

You can save up to 500 work offsets in PathPilot. The naming structure varies based on the offset number, as detailed in the following table.

Work Offset Naming		
Offsets 1-9 (Use either name)		
Offset	Extended Name	Name
1	G54.1 P1	G54
2	G54.1 P2	G55
3	G54.1 P3	G56
4	G54.1 P4	G57
5	G54.1 P5	G58
6	G54.1 P6	G59
7	G54.1 P7	G59.1
8	G54.1 P8	G59.2
9	G54.1 P9	G59.3
Offsets 10-500 (Use extended name)		
Offset	Extended Name	Name
10	G54.1 P10	Not used
11	G54.1 P11	Not used
...		
499	G54.1 P499	Not used
500	G54.1 P500	Not used

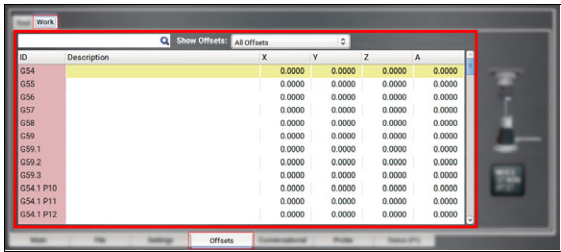
7.3.2 View Work Offsets

To view the current work offset:

- From the **Offsets** tab, on the **Work** tab, identify the **Work Offsets Table** window.

7: PATHPILOT TOOLS AND FEATURES

7.3 Set Up G-Code Programs



ID	Description	X	Y	Z	A
G54		0.0000	0.0000	0.0000	0.0000
G55		0.0000	0.0000	0.0000	0.0000
G56		0.0000	0.0000	0.0000	0.0000
G57		0.0000	0.0000	0.0000	0.0000
G58		0.0000	0.0000	0.0000	0.0000
G59		0.0000	0.0000	0.0000	0.0000
G59.1		0.0000	0.0000	0.0000	0.0000
G59.2		0.0000	0.0000	0.0000	0.0000
G59.3		0.0000	0.0000	0.0000	0.0000
G54.1 P10		0.0000	0.0000	0.0000	0.0000
G54.1 P11		0.0000	0.0000	0.0000	0.0000
G54.1 P12		0.0000	0.0000	0.0000	0.0000

Figure 7-36: Work Offsets Table window.

The active work offset is highlighted.

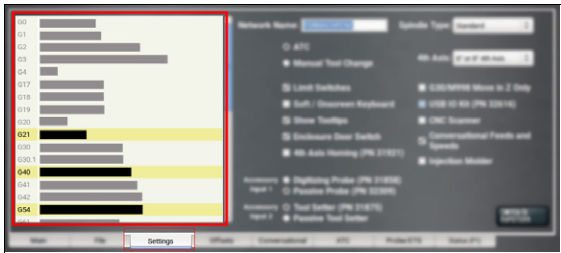
To change the current work offset, go to "Set Work Offsets" (on the previous page).

7.3.3 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.



G-Code	Description
G0	Linear Feed
G1	Linear Feed
G2	Linear Feed
G3	Linear Feed
G4	Linear Feed
G17	Linear Feed
G18	Linear Feed
G19	Linear Feed
G20	Linear Feed
G21	Linear Feed
G22	Linear Feed
G23	Linear Feed
G24	Linear Feed
G25	Linear Feed
G26	Linear Feed
G27	Linear Feed
G28	Linear Feed
G29	Linear Feed
G30	Linear Feed
G31	Linear Feed
G32	Linear Feed
G33	Linear Feed
G34	Linear Feed
G35	Linear Feed
G36	Linear Feed
G37	Linear Feed
G38	Linear Feed
G39	Linear Feed
G40	Linear Feed
G41	Linear Feed
G42	Linear Feed
G43	Linear Feed
G44	Linear Feed
G45	Linear Feed
G46	Linear Feed
G47	Linear Feed
G48	Linear Feed
G49	Linear Feed
G50	Linear Feed
G51	Linear Feed
G52	Linear Feed
G53	Linear Feed
G54	Linear Feed
G55	Linear Feed
G56	Linear Feed
G57	Linear Feed
G58	Linear Feed
G59	Linear Feed
G60	Linear Feed
G61	Linear Feed
G62	Linear Feed
G63	Linear Feed
G64	Linear Feed
G65	Linear Feed
G66	Linear Feed
G67	Linear Feed
G68	Linear Feed
G69	Linear Feed
G70	Linear Feed
G71	Linear Feed
G72	Linear Feed
G73	Linear Feed
G74	Linear Feed
G75	Linear Feed
G76	Linear Feed
G77	Linear Feed
G78	Linear Feed
G79	Linear Feed
G80	Linear Feed
G81	Linear Feed
G82	Linear Feed
G83	Linear Feed
G84	Linear Feed
G85	Linear Feed
G86	Linear Feed
G87	Linear Feed
G88	Linear Feed
G89	Linear Feed
G90	Linear Feed
G91	Linear Feed
G92	Linear Feed
G93	Linear Feed
G94	Linear Feed
G95	Linear Feed
G96	Linear Feed
G97	Linear Feed
G98	Linear Feed
G99	Linear Feed
G100	Linear Feed

Figure 7-37: G-code Description window on the Settings tab.

7.4 RUN G-CODE PROGRAMS

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

7.4.1 Bring the Machine Out of Reset	107
7.4.2 View the Active Axis to Jog	107
7.4.3 Jog the Machine	107
7.4.4 View the Current Machine Position	108
7.4.5 Reference the Machine	108
7.4.6 Start a Program	109
7.4.7 Stop Machine Motion	109
7.4.8 View the Active G-Code Modes	109
7.4.9 View the Distance to Go	110

7.4.1 Bring the Machine Out of Reset

- Select **Reset**.

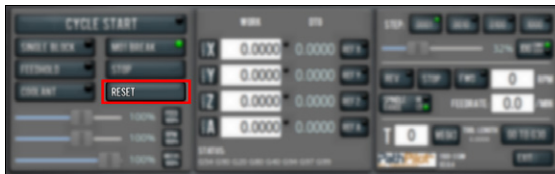


Figure 7-38: Reset button.

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

About Reset Mode

When the machine is first powered on, or after an emergency stop, the Reset button flashes. When you select the flashing Reset button, PathPilot verifies communication to the machine and does the following activities:

- Brings the machine out of an emergency stop condition
- Clears alarms
- Clears the tool path backplot
- Resets all modal G-codes to their normal state
- Rewinds the currently loaded G-code program
- Stops machine motion, but is **not** a replacement for the Emergency Stop button

You can select the Reset button any time while the machine is on.

7.4.2 View the Active Axis to Jog

To find which axis is active while jogging your machine:

- Identify the light next to the **Work Offset** DRO fields.

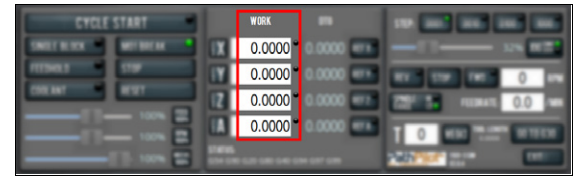


Figure 7-39: Work Offset DRO fields.

For information, see "Jog the Machine" (below).

7.4.3 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.

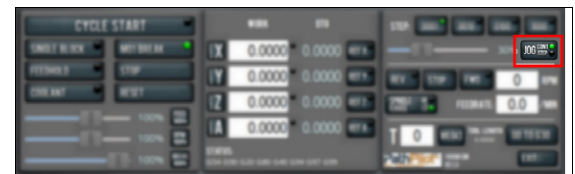


Figure 7-40: Jog button.

When the **Cont** green light is on, continuous velocity mode is selected.

When the **Step** green light is on, step mode is selected.

To use continuous velocity mode:

- Set the velocity: drag the **Jog Speed** slider.

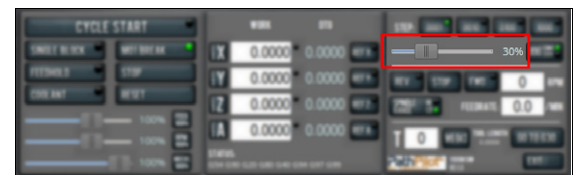


Figure 7-41: Jog Speed slider.

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

To use step mode, select the step size. Do one of the following, depending on your accessories:

- In the **Manual Control Area**, in the **Jog** group, select the step size.

The **Step** button's light comes on, indicating which step size is active.



Figure 7-42: Step buttons (in G20 mode).

7: PATHPILOT TOOLS AND FEATURES

7.4 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" (below).

Jog in Continuous Velocity Mode

In continuous mode, the machine jogs at a continuous velocity.

To select continuous velocity mode:

- In the **Manual Control** area, select **Jog**.

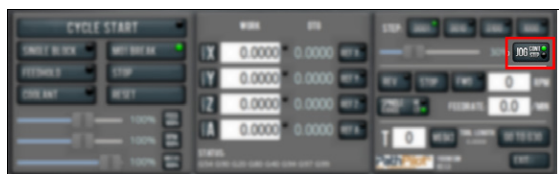


Figure 7-43: Continuous velocity jogging controls.

When the **Cont** green light is on, continuous velocity mode is selected.

When the **Step** green light is on, step mode is selected.

To set the velocity:

- Drag the **Jog Speed** slider.

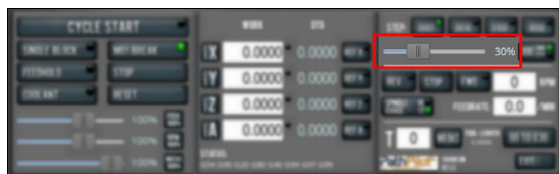


Figure 7-44: Jog Speed slider.

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

This is useful when you're doing things like:

- Roughly positioning the machine (for example, to move the spindle head away from the workpiece).
- Moving the machine a certain distance at a constant speed.

Jog in Step Mode

In step mode, the machine jogs in steps, which range based on the programming mode you're using:

- Imperial (G20) Mode** 0.0001 in. to 0.1000 in.
- Metric (G21) Mode** 0.01 mm to 10 mm

To select the step size:

- In the **Manual Control Area**, select the step size.
The **Step** button's light comes on, indicating which step size is active.

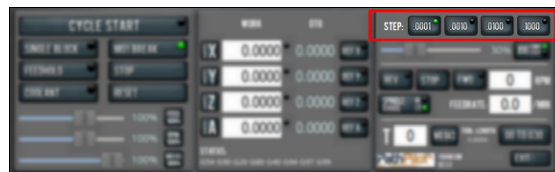


Figure 7-45: Step buttons (in G20 mode).

About Step Jogging

While jogging in step mode, the machine moves one step each time you either press a jog key on the keyboard or click the inner wheel of the Jog Shuttle. The jog step sizes range depending on the programming mode you are using:

- Imperial (G20) Mode** 0.0001 in. to 0.1000 in.
- Metric (G21) Mode** 0.01 mm to 10 mm

Step jogging mode is useful to finely move the machine, like when you're indicating a workpiece or manually setting tool lengths.

The jog keys on the keyboard only move the machine in steps when step mode is indicated in PathPilot. The inner wheel on the jog shuttle always moves the machine in steps, regardless of which mode is indicated in PathPilot.

7.4.4 View the Current Machine Position

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

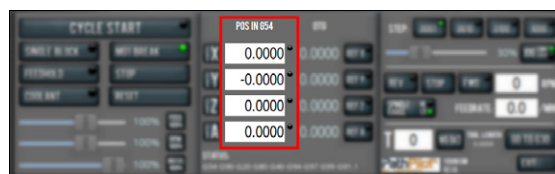


Figure 7-46: Work Offset DRO fields.

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 105).

7.4.5 Reference the Machine

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

- To verify that the tooling is clear of any possible obstructions, reference the Z-axis before referencing the other axes: from the PathPilot interface, select **Ref Z**.



Figure 7-47: Reference buttons.

- Once the spindle is clear of any possible obstructions, continue referencing all axes.



Note: You can select the buttons one after another. Once the machine references one axis, it'll move on to the next.

After each axis is referenced, its button light comes on. For more information on referencing the machine, see "About Referencing" (below).

About Referencing

You must reference the machine to establish a known position for PathPilot. The position that's set while referencing the machine is the origin of the machine coordinate system. Without referencing the machine, PathPilot won't know the current position of the machine axes.

You must reference the machine at the following times:

- After you power on the machine
- After you push in the Emergency Stop button
- Before running a G-code program
- Before using MDI commands
- Before setting work or tool offsets
- After a collision or an axis stall/fault

When referencing, the machine moves each axis to the end of its travel. The machine stops at the limit switch, which sets the axis' reference position.

7.4.6 Start a Program

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

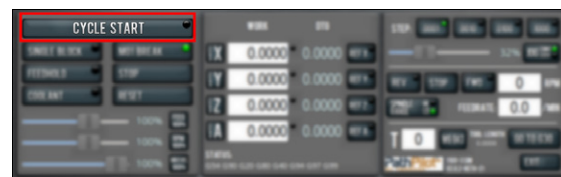


Figure 7-48: Cycle Start button.

For more information on starting a program, see "About Cycle Start" (below).

If you can't start a program, go to "Cycle Start Reference" (below).

About Cycle Start

While a program is running, the Cycle Start button's light is on. The Cycle Start button's light flashes if motion is paused during the program. The following modes may pause motion during a program:

- Single block
- Feed hold
- M01 break

If machine motion pauses a single block, feed hold, or M01 break, the Cycle Start button flashes until it's selected again.

Cycle Start Reference

The Cycle Start button doesn't operate if you select it:

- While you're not in the Main tab. For information, see "Main Tab" (page 74).
- Before you've loaded a G-code program. For information, see "Load G-Code" (page 88).
- Before referencing the machine. For information, see "Reference the Machine" (on the previous page).

7.4.7 Stop Machine Motion

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



Figure 7-49: Stop button.

7.4.8 View the Active G-Code Modes

To find the currently active G-code modes and the currently active tool at a glance:

7: PATHPILOT TOOLS AND FEATURES

7.4 Run G-Code Programs

- Identify the **Status** read-only DRO field.



Figure 7-50: Status read-only DRO field.

For more information on G-code modes, go to "View Available G-Code Modes" (page 106).

7.4.9 View the Distance to Go

To view the distance to go:

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

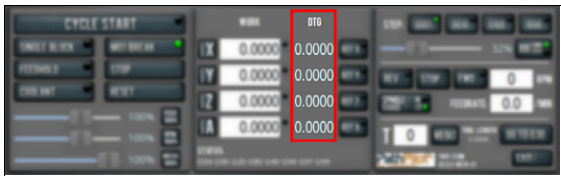


Figure 7-51: DTG read-only DRO fields.

The value is the remaining distance in any programmed move.

For more information on using the **DTG** read-only DRO fields, see "About Distance to Go" (below).

About Distance to Go

While a program is running, the DTG read-only DRO fields show the remaining distance in each move.

After using the feed hold function or the maxvel override function, look at the distance to go. This read-only DRO field is useful to prove out a part program.

7.5 CONTROL G-CODE PROGRAMS

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

7.5.1 Use the Feed Hold Function	111
7.5.2 Use the Feed Rate Override Function	111
7.5.3 Use M01 Break Mode	112
7.5.4 Use the Maxvel Override Function	112
7.5.5 Use Single Block Mode	112
7.5.6 Use the Voltage Override Function	112
7.5.7 Change the Tool Number	113
7.5.8 Use a G30 Position	113
7.5.9 Manually Enter Commands	113
7.5.10 Copy Recently Entered Commands	115
7.5.11 Use the AutoFS Material Picker	115
7.5.12 Use Cycle Counters (M30 and M99)	116

7.5.1 Use the Feed Hold Function

- Select **Feed Hold**.



Figure 7-52: 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 109).



Note: If the machine isn't moving, the feed hold function doesn't have an effect.

You can use the feed hold function either while a program is running or while you are using manual data input (MDI) commands. If the program is running a spindle-synchronized move, the feed hold function is delayed until the move is complete.

7.5.2 Use the Feed Rate Override Function

To use the feed rate override function:

- Using the **Feed Rate Override** slider, change the programmed feed rate by a specific percentage.



Figure 7-53: Feed Rate Override slider.



Note: Percentages range from 1-200%.

To remove the feed rate override function:

- Select **Feed 100%**.
The feed rate returns to 100% of its programmed value (it's no longer overridden).

For more information on the feed rate override function, see "About Feed Rate Override" (below).

About Feed Rate Override

You can use the feed rate override function while you're doing any of the following activities:

- Using manual data input (MDI) commands
- Jogging
- Running a program with G01, G02, or G03 commands

The feed rate override function does not affect G00 (rapid) commands. It's ignored if:

- The program is running a spindle-synchronized move
- An M48 (disable feed and speed overrides) command is used

To indicate lack of motion or unusual levels, the slider turns yellow when it's either at 0% or above 100%.

The Feed Rate Override slider and Feed 100% button work similarly to the spindle override controls. They affect the commanded feed rate by a percentage from 1-200%. The feed rate override works for MDI, jogging, and G-code program G01/G02/G03 moves. The override has no effect on G00 (rapid) moves.

7: PATHPILOT TOOLS AND FEATURES

7.5 Control G-Code Programs

7.5.3 Use M01 Break Mode

- Select **M01 Break**.

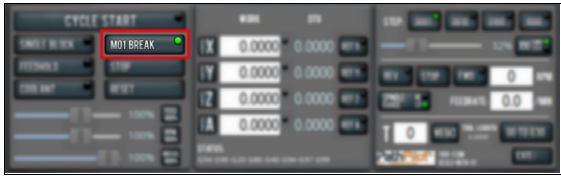


Figure 7-54: M01 Break button.

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 109).
- **When M01 Break is Inactive** PathPilot ignores all programmed M01 commands.

7.5.4 Use the Maxvel Override Function

To use the maxvel override function:

- Using the **Maxvel Override** slider, change the maximum velocity by a specified percentage.



Figure 7-55: Maxvel Override slider.

To remove the maxvel override function:

- Select **Maxvel 100%**.

For more information on using the maxvel override function, see "About Maxvel Override" (below).

About Maxvel Override

The maxvel override function affects G00 and G01 commands, and it's useful for:

- **Running a Program for the First Time** Drag the Maxvel Override slider to 0% to verify that all DRO fields look appropriate.
- **Safety** If you're running a spindle-synchronized move, a maxvel override isn't ignored. Verify that the maxvel override value allows the machine to use the programmed feed rate during spindle-synchronized moves. If it can't, the spindle-synchronized move won't produce the results you want.

To indicate lack of motion or unusual levels, the slider turns yellow when it's either at 0% or above 100%.

7.5.5 Use Single Block Mode

- Select **Single Block**.



Figure 7-56: 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 109).

You can turn single block mode on or off either before starting a program or while a program is running. For information, see "Use Single Block Mode" (above).



Note: Single block mode ignores non-motion lines, like comment lines or blank lines.

7.5.6 Use the Voltage Override Function

To use the voltage override function:

- Using the **Voltage Override** slider, change the programmed voltage by a specific percentage.

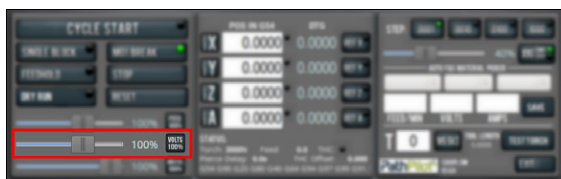


Figure 7-57: Voltage Override slider.



Note: Percentages range from 1-200%.

To remove the voltage override function:

- Select **Volts 100%**.
The voltage returns to 100% of its programmed value (it's no longer overridden).

7.5.7 Change the Tool Number

The Tool DRO field shows the tool number currently active.

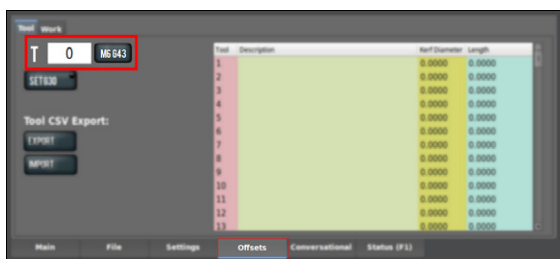


Figure 7-58: Tool DRO field.

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

1. From the PathPilot interface, on the **Offsets** tab, select the **Tool** tab.
2. In the **Tool DRO** field on the **Offsets** tab, type a number (the valid range is from 0-1000). Then select the **Enter** key.



Note: You can also select **M6 G43**. For information, see "About M6 G43" (below).

About M6 G43

The M6 G43 button is a shortcut used to do the following:

- Change the number of the currently-loaded tool in the spindle to the number typed in the Tool DRO field. This is the equivalent of an M06 command.

7.5.8 Use a G30 Position

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

To set a G30 position:

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

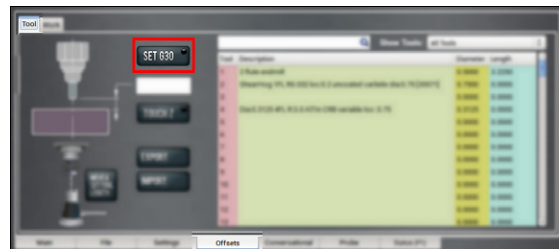


Figure 7-59: Set G30 button.

To go to a set G30 position:

- Use a **G30** command in a G-code program.



Note: The G30 position defaults to only moving the Z-axis.

About G30

A G30 command in a G-code program moves the machine to a preset position. For more information on setting a G30 position, see "Use a G30 Position" (above).

Use a G30 move to start a coordinated movement of the axes. You can limit the movement to only the Z-axis. For information, see "Limit G30 Moves" (page 100).



Tip! It's useful to program a G30 move right before a tool change so that the machine can jog to a safe tool change position.

7.5.9 Manually Enter Commands

You can send G-code commands directly to the machine by using the MDI Line DRO field. For information, see "About the MDI Line DRO Field" (on the next page).

To manually enter commands:

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

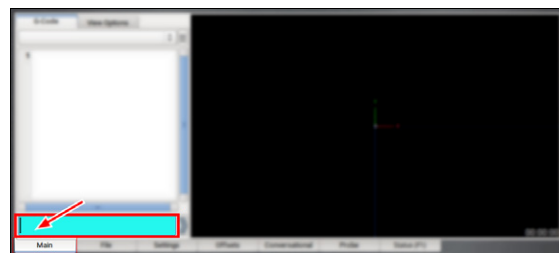


Figure 7-60: MDI Line DRO field.

The DRO field highlights.

7: PATHPILOT TOOLS AND FEATURES

7.5 Control G-Code Programs

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" (on the previous page).

The MDI Line DRO field saves up to 100 of your most recent commands, which are saved after a power cycle.

When you select the MDI Line DRO field, all keystrokes are used within the field — so, you can't jog the machine.

Admin Commands Reference

Use the following commands in PathPilot:

Admin Command	Use to...
ADMIN AUDIO	Customize the controller's audio device settings.
ADMIN CALC	Open the calculator.
ADMIN CLEAR	Clear the message history on the Status tab.
ADMIN CONFIG	Change the configuration of the PathPilot interface.
ADMIN CYCLECOUNTER	Control the cycle counters in the Tool Path display.
ADMIN DATE	Customize the controller's date and time.
ADMIN DISPLAY	Customize the controller's screen display.
ADMIN DROPBOX	Connect your controller to a Dropbox account for cloud file syncing.
ADMIN HELP	Review a list of available Admin commands.
ADMIN KEYBOARD	Customize the controller's keyboard layout.

Admin Command	Use to...
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.
ADMIN SET_Y_LIMIT	Set the Y-axis soft limit.
ADMIN SET_Z_LIMIT	Set the Z-axis soft limit.
ADMIN SETTINGS BACKUP	Create a backup of tool offset and fixture information to store externally.
ADMIN SETTINGS RESTORE	Restore tool offset and fixture information backup from an external location.
ADMIN SHOW_SOFT_LIMITS	Display the current axis soft limits on the Status tab.
ADMIN TOOLTIP DELAYMS	Set the milliseconds prior to displaying the tooltip (and then again for the expanded tooltip). The default is 1200 milliseconds.
ADMIN TOOLTIP MAXDISPLAYSEC	Limit the amount of time the expanded tooltip displays. The default is 15 seconds.
ADMIN TOUCHSCREEN	Adjust the touch screen calibration.
ADMIN VERSION	Display detailed version information on the Status tab.

7.5.10 Copy Recently Entered Commands

1. From the **MDI Line** DRO field, press either the **Up Arrow** key or the **Down Arrow** key.
The previously entered command displays.
2. You must press the **Enter** key to execute the command.
To abandon the command, press **Esc**.

For information, see "Manually Enter Commands" (page 113).

7.5.11 Use the AutoFS Material Picker

PathPilot on the 1300PL is designed so that you don't need to manually program feeds and speeds in your G-code. Instead, feed rate, amperage, pierce parameters, and torch height control voltage are all set at program run-time using automatic look-up tables based on the material you're cutting. AutoFS is designed so that you can export a program once from your post-processor and run it multiple times with different materials, all without re-posting. All material-specific machine parameters are set by the M200 AutoFS M-code (rather than directly programmed in the G-code).

Writing G-Code with AutoFS

A standard block of G-code before a cutting operation might look like this:

```
M210 P123 (Set THC voltage to 123v)
M211 P45 (Set plasma source to 45A)
M207 P0.15 (Set pierce height to 0.15")
M208 P0.08 (Set cut height to 0.08")
M209 P2 (Set a two second pierce delay)
F225 (Set cutting feed rate of 225 IPM)

G15 (Perform ohmic probe)
G16 (Pierce and start cutting)
```

The block of code shown above is totally valid and can be used if you prefer to hard code your cutting parameters.

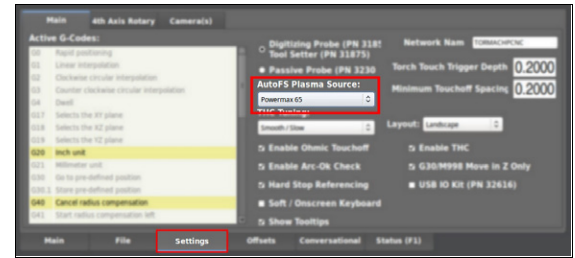
Alternatively, the code shown above can be replaced with the following when AutoFS is being used:

```
M200 (Set cutting parameters from AutoFS tables)

G15 (Perform ohmic probe)
G16 (Pierce and start cutting)
```

Using AutoFS

1. On the **Settings** tab, select which plasma source you're using from the **Auto FS Plasma Source** drop-down menu.



2. Before running your program, select the material you're cutting from the AutoFS dropdowns on the **Main** tab of PathPilot.

The DROs turn green, which indicates that they were automatically set.

With a material selected, the next time an **M200** code is encountered in a program, PathPilot will set all of the cutting parameters for that material.

Creating Custom AutoFS Presets

Since your shop air supply and cutting conditions might differ from those used to create the AutoFS presets, you can save your own custom settings for a material at any time.

To create new AutoFS presets:

1. From the **Main** tab, select the material you're cutting from the AutoFS dropdowns, and then type your cut parameters into the DRO fields.
2. Select **Save**.
The values are stored as a new preset for that material.

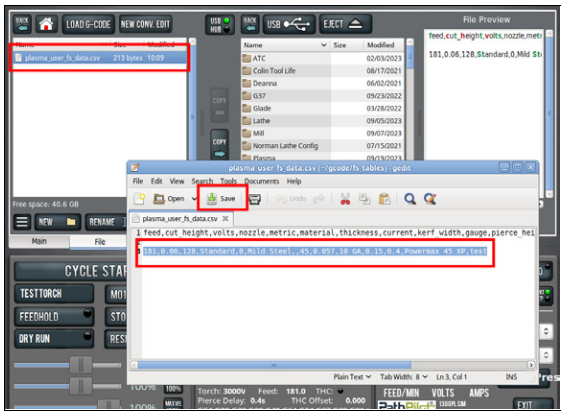
Deleting Custom AutoFS Presets

1. From the **File** tab, navigate to the **fs_tables** folder, and open the **plasma_user_fs_data.csv** file.

7: PATHPILOT TOOLS AND FEATURES

7.5 Control G-Code Programs

2. Highlight the line of the preset you want to delete and delete it. Then, save the file.



Note: This file is read on boot only, so the preset remains visible until the controller is power cycled.

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.

7.5.12 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 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

7.6 SYSTEM FILE MANAGEMENT

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

7.6.1 Manage System Files	117
7.6.2 Create Backup Files	117
7.6.3 Restore Backup Files	118
7.6.4 Import and Export the Tool Table	119

7.6.1 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 88).

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**.

7.6.2 Create Backup Files

1. Insert a blank, formatted USB drive into the PathPilot controller.

Note: To prevent errors when backing up and restoring files, only use a blank, formatted USB drive.

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

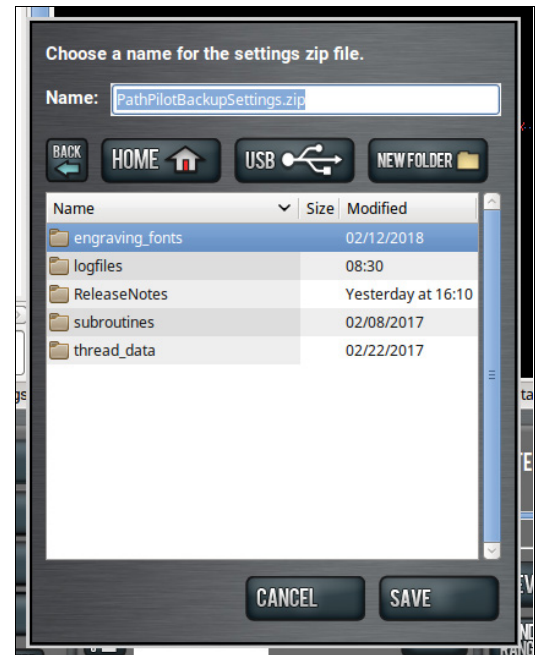


Figure 7-61: Admin Settings Backup dialog box.

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 7-62: 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.

7: PATHPILOT TOOLS AND FEATURES

7.6 System File Management

6. To prevent errors, make sure you don't include the following folders:
- **logfiles**
 - **media**
 - **ReleaseNotes**
 - **subroutines**
 - **USB**
7. Select **Copy to USB**.
- The files are copied and display in the **USB Files** window.
8. Eject the USB drive from the PathPilot controller.
9. From the PathPilot interface, select **Exit**.
10. Verify that all files are properly saved: insert the USB drive on a device other than the PathPilot controller, and review the list of files on the USB drive.
11. (Optional) As an extra precaution, copy all the files onto the device.

About Backup Files

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

7.6.3 Restore Backup Files

1. Insert the USB drive with your backup files into the PathPilot controller.
 2. From the PathPilot interface, on the **Main** tab, in the **MDI Line DRO** field, type **ADMIN SETTINGS RESTORE**. Then select the **Enter** key.
- The **Admin Settings Restore** dialog box displays.

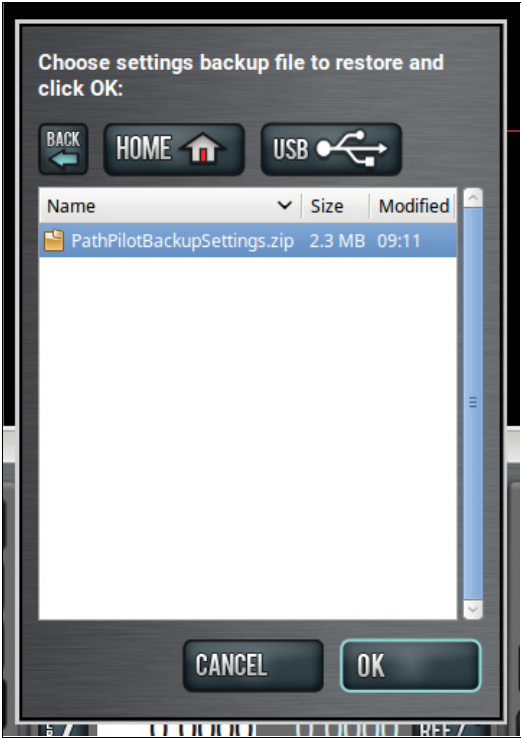


Figure 7-63: Admin Settings Restore dialog box.

3. From the **Admin Settings Restore** dialog box, navigate to the backup .zip file on the USB drive, and then select **OK**. The PathPilot operating system restores the backup, then restarts.
4. If you backed up any other files onto the USB drive, you must manually transfer the files to the PathPilot controller. From the PathPilot interface, on the **File** tab, in the **USB Files** window, select the files you want to transfer.

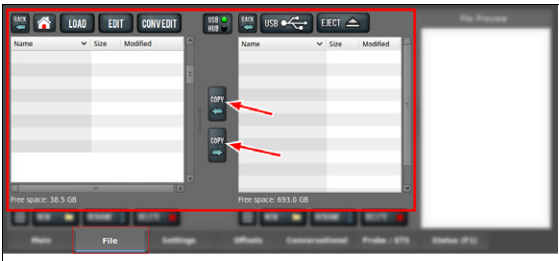


Figure 7-64: USB Files window on the File tab.

 **Note:** To navigate backward, select **Back**. To navigate to the top level, select **USB**.

5. 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.

7.6.4 Import and Export the Tool Table

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

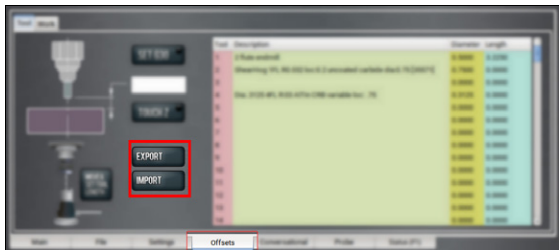


Figure 7-65: Export and Import buttons on the Offsets tab.

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, select **Import**.

The **Import** dialog box displays.

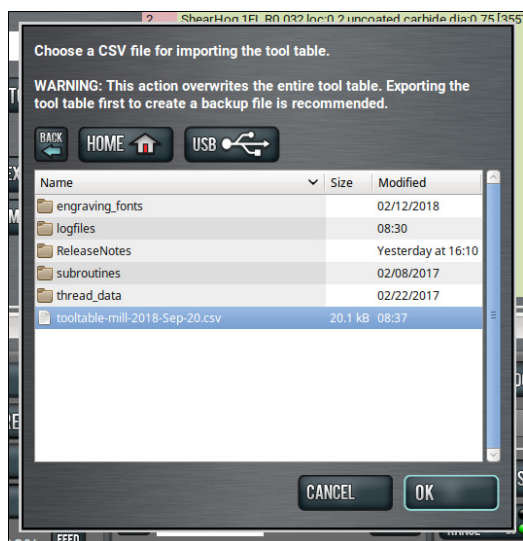


Figure 7-66: Import dialog box.

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, select **Export**.

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

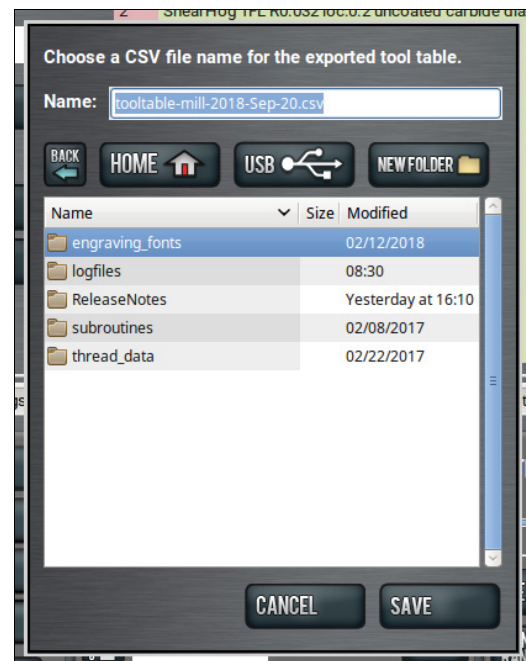


Figure 7-67: Export dialog box.

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.

TORMACH



BASIC OPERATIONS

IN THIS SECTION, YOU'LL LEARN:

- About the basic operations required for most projects, organized as a suggested project workflow.

CONTENTS

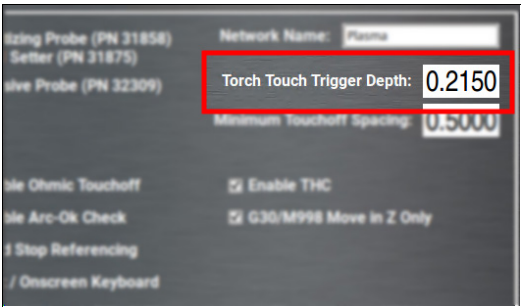
8.1 Plasma Settings Reference.....	122
8.2 Create Programs with CAD/CAM.....	124
8.3 Select a Method for Initial Height Sensing (IHS).....	125
8.4 Make Your First Cuts.....	126
8.5 Use the AutoFS Material Picker.....	128

8: BASIC OPERATIONS

8.1 Plasma Settings Reference

8.1 PLASMA SETTINGS REFERENCE

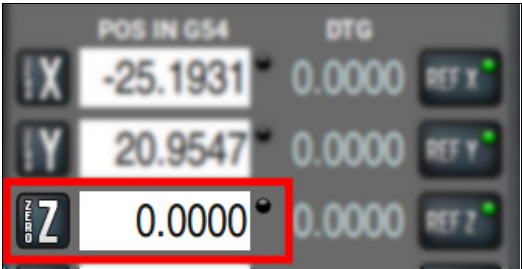
8.1.1 Touch Switch Trigger Depth



When using the physical touchoff switch for material height sensing, there's a certain amount of lost motion in the Z-axis between when the torch head touches the material and when the micro-switch in the torch lifter triggers. It's important to set an accurate value for the distance between torch touch and touch switch triggering so that the initial pierce height can be correctly computed.

To set torch touch trigger depth:

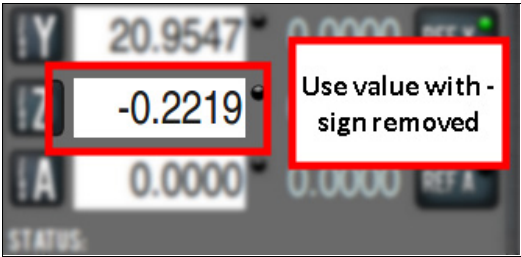
- 1. Power on the machine and reference all axes.
- 2. From the PathPilot interface, on the **Settings** tab, disable **Ohmic Touchoff**.
- 3. Put a piece of material on the machine table and position the torch above it.
- 4. Using a slow jog speed (~5 IPM), jog the torch down until it's just contacting the material.
- 5. Zero your Z-axis at this position.



- 6. Switch to the **Status** tab and watch the **Touch Switch** LED. Very slowly, jog downwards until the LED comes on.

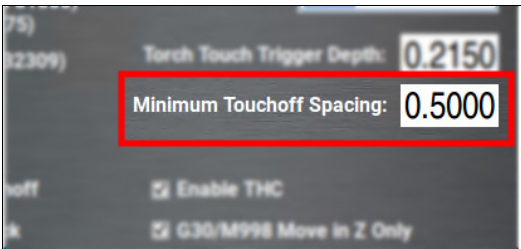


- 7. Note the value in the **Z DRO** field. This is your Torch Touch Trigger Depth. Convert this value to a positive number and enter it on the **Settings** tab.



- 8. Re-enable **Ohmic Touchoff** on the **Settings** page.

8.1.2 Minimum Touchoff Spacing



Minimum touchoff spacing is a time saving setting designed to avoid unnecessary probing during programs with a large number of individual cuts.

After the machine has probed for initial cut height, if the next cut begins within this distance it will be assumed that the material height is the same. This allows the machine to skip the probing routine and proceed straight to pierce height. If your material is very flat and uniform, you can use a large distance for this setting (>2 in.). If your material is uneven or you would simply prefer to probe for height every time a cut begins, enter zero for this value.

8.1.3 Enable Ohmic Touchoff

This setting disables the ohmic sensing system entirely and relies solely on the physical touch switch for height sensing. This can be useful for cutting programs that result in water being repeatedly splashed into the torch nozzle, causing spurious probing trips.



Note: Do not disable ohmic touch-off when cutting very thin material. The weight of the torch will bend the material during probing and cause inaccurate pierce height.

8.1.4 Enable THC

Disabling this setting stops all dynamic height adjustment during cutting. The torch will stay at the same height for the length of a cut and not adjust to follow the contours of the material.

8.1.5 Enable Arc-Ok Checking

Hypertherm plasma sources supply an arc-ok signal to PathPilot when the cutting arc is fully established. When this setting is enabled, the machine will pause during a G16 pierce until it receives an arc-ok signal.

Disable this setting if your plasma source does not supply an arc-ok signal.

8: BASIC OPERATIONS

8.2 Create Programs with CAD/CAM

8.2 CREATE PROGRAMS WITH CAD/CAM

8.2.1 Converting DXF Files to G-Code

The simplest way to start cutting on the 1300PL is by using the conversational DXF import feature in PathPilot. We recommend this for beginners because it ensures that your G-code takes full advantage of PathPilot's AutoFS features and integrates the recommended feeds and speeds for Hypertherm plasma units.

To create DXF files for cutting:

1. Model the part as a flat body in SolidWorks. Use the sheet metal design tools or a extrude a flat sketch to create the geometry.
2. Export the DXF by right-clicking the desired surface in SolidWorks and selecting **Export to DXF/DWG**.

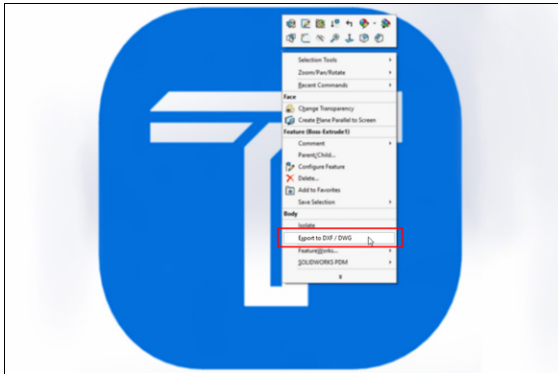


Figure 8-1: Example of exporting a surface in SolidWorks to DXF/DWG.

This approach avoids including unwanted items like axis markings, sketch boundaries, or notes in the DXF file.

3. Once your DXF is ready, import it into PathPilot using the conversational DXF import tool to quickly generate reliable cutting programs.

8.2.2 Using a Post-Processor

For more control over your cutting programs, you can use one of the following CAM programs, which have Tormach-supported post-processors:

- Autodesk Fusion 360
- Vectric Software

To export programs using Fusion or Vectric:

1. Select the Tormach post-processor in your CAM software.

2. Choose whether to:

- Use the M200 AutoFS feature that allows PathPilot to choose feeds and speeds (based on the AutoFS dropdowns).
- Override the AutoFS settings by applying the cut parameters from your CAM software.



Note: If you override AutoFS settings, you may experience unexpected cutting problems. Ensure that your cut settings align with Hypertherm's recommendations.

To export programs using another CAM software:

- Modify the post-processor to ensure the generated G-code matches the requirements in the example program in "Make Your First Cuts" (page 126).



Note: Although PathPilot uses LinuxCNC, off-the-shelf post-processors will not work without adjustments. They lack the specific M- and G-codes that PathPilot uses for setting cut parameters, so using them as-is may produce unexpected results.

8.3 SELECT A METHOD FOR INITIAL HEIGHT SENSING (IHS)

Because plasma cutting workpieces are prone to heat-induced warping, it's common practice to probe the workpiece height before each cut using the initial height sensing (IHS) system. This guarantees a consistent pierce height, which prolongs consumable life.

The 1300PL has two methods of detecting the Z location of the workpiece before a cut. Choose which method to use based on your workpiece, cutting situation, and machine setup.

To select an IHS method:

- From the PathPilot interface, on the **Settings** tab, select or clear the **Enable Ohmic Touchoff** checkbox.
For more information, see "Plasma Settings Reference" (page 122).

8.3.1 Physical Touch Switch

The first (and simplest) IHS method is the physical touch off switch in the torch lifter. A spring-loaded switch on the Z lead screw is triggered when the torch is pressed against the workpiece.

Use the physical touch switch when:

- Your workpiece is thicker than 18 ga (1.2 mm).
- Making many small cuts, where water is splashed into the torch head between cuts.

8.3.2 Ohmic Probing

The second, and more accurate, method of IHS is ohmic probing. Ohmic probing uses the conductivity of the workpiece to detect when the torch cap is touching it. Ohmic probing requires a very light touch and is very accurate, but can be incorrectly triggered by water splashing into the torch head.

Use ohmic probing when:

- Your workpiece is thinner than 18 ga (1.2 mm).
- Cutting large workpieces, where the torch is not crossing over open water between cuts.

8: BASIC OPERATIONS

8.4 Make Your First Cuts

8.4 MAKE YOUR FIRST CUTS

Once you've verified all subsystems of the machine and plasma source, we recommend making a simple test program to test the cutting and torch height control systems. This is also a good quick-start tutorial on NC programming for the 1300PL.

8.4.1 Test G-Code

The following code will probe, set feeds from the AutoFS system, and then cut a 4" line from (0, 0) to (0, 4). Open a text editor on your computer (like Notepad++), and save the following code in a file named `test.nc`.

```
G90 (Absolute Distance Mode)
G64 P 0.0050 Q 0.0050 (PathBlending)
G17 (XY Plane)
G54 (Set Work Offset)
T 1 M6 G43 H 1

G30 (Move Home)
G0 X0 Y0

M200 (AutoFS)
G15 (probe)
G16 (pierce)

G1 X0 Y4

M205 (Torch Off)
G30 (Move Home)
```

8.4.2 Code Breakdown

Preamble: This code block resets the modal state of the machine. It places the interpreter in absolute distance mode, sets reasonable values for smoothing, and selects the G54 work offset. For more detail on these G-codes, see the full PathPilot manual.

```
G90 (Absolute Distance Mode)
G64 P 0.0050 Q 0.0050 (PathBlending)
G17 (XY Plane)
G54 (Set Work Offset)
```

Tool Selection: Selects Tool 1 from the offsets table. On the plasma, convention is to use Tool 1 for the plasma torch.

```
T 1 M6 G43 H 1
```

Initial Move: Moves the torch to the home (G30) position in Z for safety and then performs a rapid move to coordinate (0, 0).

```
G30 (Move Home)
G0 X0 Y0
```

AutoFS: Uses the AutoFS material selector on the Main tab of PathPilot to set feed rate, pierce delay, pierce height and plasma amperage. No explicit feedrate needs to be specified for G1 moves since AutoFS sets this for you.

```
M200 (AutoFS)
```

Probe and Pierce: Probe the material surface using ohmic sensing. Once the torch detects the surface, Z0 for the current Work Coordinate System is set at the surface. After probing, G16 moves the torch to the pierce height set by AutoFS, fires the torch, waits for the AutoFS pierce delay and then moves to cutting height.

```
G15 (probe)
G16 (pierce)
```

Cut and Turn Torch Off: Performs a feed move from the current position (0, 0) to (0, 4). After the cutting move is complete, the torch is shut off and returned to the G30 home position.



Note: You might notice that your torch pauses and continues to blow air for 10-20 seconds. This is a cool down procedure designed by the plasma source manufacturer.

```
G1 X0 Y4
M205 (Torch Off)
G30 (Move Home)
```

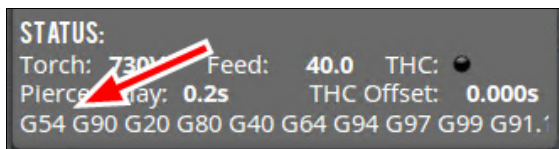
8.4.3 Run the Test Program



IMPORTANT! Before running your test program, make sure you have completed all steps in "Verify the Installation" (page 56).

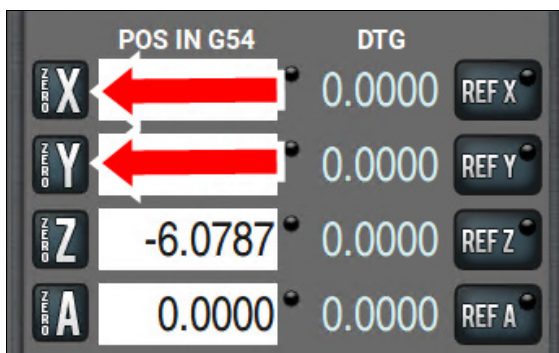
1. Find a piece of scrap sheet steel large enough to make a 4" test cut.
2. Put the metal on the cutting table and connect the ground clamp to it.
3. Bring PathPilot out of the Reset state and home the machine.

4. Power on plasma source.
5. Jog the cutting torch to a place at least 5" past the edge of the plate in Y- (towards the operator side of the machine).
6. Check that your current Work Coordinate System is G54 to match the test cutting code.

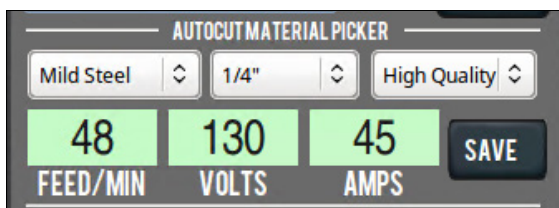


If it isn't, type G54 into the **MDI Line DRO** field. Then select the **Enter** key.

7. Select **Zero X** and **Zero Y** to set your work coordinate system zeros.



8. Copy your G-code from a USB drive and load it into PathPilot.
9. Select the material you're using for your cut test in the AutoFS dropdown on the **Main** tab.



10. Adjust your plasma source to the amperage shown in the **AMPS** DRO field.
11. Select **Cycle Start**.
The machine performs the test cut that you programmed.

8: BASIC OPERATIONS
8.5 Use the AutoFS Material Picker

8.5 USE THE AUTOFS MATERIAL PICKER

PathPilot on the 1300PL is designed so that you don't need to manually program feeds and speeds in your G-code. Instead, feed rate, amperage, pierce parameters, and torch height control voltage are all set at program run-time using automatic look-up tables based on the material you're cutting. AutoFS is designed so that you can export a program once from your post-processor and run it multiple times with different materials, all without re-posting. All material-specific machine parameters are set by the M200 AutoFS M-code (rather than directly programmed in the G-code).

8.5.1 Writing G-Code with AutoFS

A standard block of G-code before a cutting operation might look like this:

```
M210 P123 (Set THC voltage to 123v)
M211 P45 (Set plasma source to 45A)
M207 P0.15 (Set pierce height to 0.15")
M208 P0.08 (Set cut height to 0.08")
M209 P2 (Set a two second pierce delay)
F225 (Set cutting feed rate of 225 IPM)

G15 (Perform ohmic probe)
G16 (Pierce and start cutting)
```

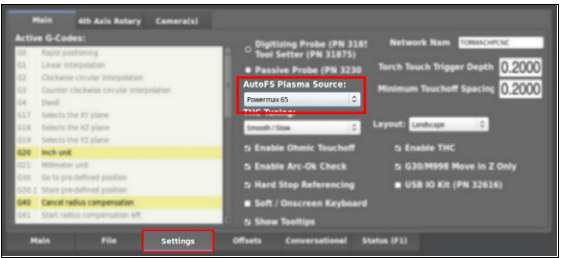
The block of code shown above is totally valid and can be used if you prefer to hard code your cutting parameters. Alternatively, the code shown above can be replaced with the following when AutoFS is being used:

```
M200 (Set cutting parameters from AutoFS tables)

G15 (Perform ohmic probe)
G16 (Pierce and start cutting)
```

8.5.2 Using AutoFS

- 1. On the Settings tab, select which plasma source you're using from the Auto FS Plasma Source drop-down menu.



- 2. Before running your program, select the material you're

cutting from the AutoFS dropdowns on the Main tab of PathPilot. The DROs turn green, which indicates that they were automatically set.

With a material selected, the next time an M200 code is encountered in a program, PathPilot will set all of the cutting parameters for that material.

8.5.3 Creating Custom AutoFS Presets

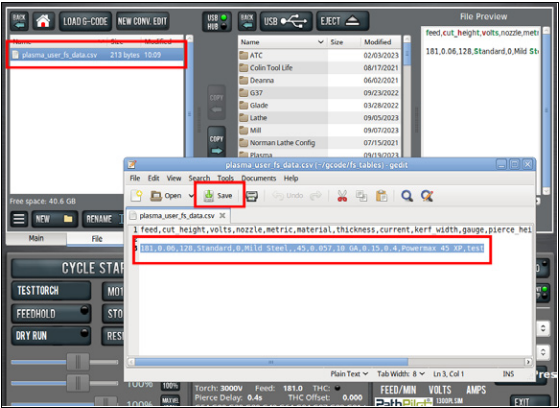
Since your shop air supply and cutting conditions might differ from those used to create the AutoFS presets, you can save your own custom settings for a material at any time.

To create new AutoFS presets:

- 1. From the Main tab, select the material you're cutting from the AutoFS dropdowns, and then type your cut parameters into the DRO fields.
- 2. Select Save. The values are stored as a new preset for that material.

8.5.4 Deleting Custom AutoFS Presets

- 1. From the File tab, navigate to the fs_tables folder, and open the plasma_user_fs_data.csv file.
- 2. Highlight the line of the preset you want to delete and delete it. Then, save the file.



Note: This file is read on boot only, so the preset remains visible until the controller is power cycled.

PROGRAMMING

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

9.1 Before You Begin	130
9.2 Programming Overview	131
9.3 Programming G-Code	137
9.4 Programming M-Code	149
9.5 Programming Input Codes	154
9.6 Advanced Programming	155

9: PROGRAMMING

9.1 Before You Begin

9.1 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.

9.2 PROGRAMMING OVERVIEW

Read the following sections for a G-code overview:

9.2.1 About G-Code Programming Language	131
9.2.2 G-Code Formatting Reference	131
9.2.3 Supported G-Codes Reference	135

9.2.1 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).

9.2.2 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)

Programs are limited to 999,999 lines of code.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. Blank lines are allowed in the input, but they're ignored. Input is not case sensitive (except in comments), so any letter outside a comment may be in uppercase or lowercase without changing the meaning of a line.



EXAMPLE

G00 x +0.12 34y 7 is equal to G00 x+0.1234 y7

A line may have:

- Any number of G words, but two G words from the same modal group may not appear on the same line.
- Zero to four M words, but two M words from the same modal group may not appear on the same line.
- For all other legal letters, a line may have only one word beginning with that letter.

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

PathPilot omits blocks of code that are prefixed with a block delete character (/).

PathPilot sometimes ignores things it doesn't understand. If a command doesn't work as expected, or does nothing, make sure that it's correctly typed. PathPilot doesn't check for excessively high machining feeds or speeds, and it doesn't detect situations where a legal command will do something unfortunate (like machining a fixture).

Line Numbers

A line number is indicated by the following, in the order listed:

1. The letter N
2. An integer (with no sign) between 0 and 99,999,999 (which must be written without commas)

Line numbers may be repeated, or used out of order, but that's rare in normal practice. A line number isn't required, and is often omitted.

Words

A word is indicated by the following, in the order listed:

9: PROGRAMMING

9.2 Programming Overview

1. A letter other than N or O
2. A real value

Letters

Words may begin with any of the following letters, except N or O:



Note: Several letters (I, J, K, L, P and R) may have different meanings in different contexts.

Letter	Description
A	A-axis
B	B-axis
C	C-axis
D	Tool radius compensation number
F	Feed rate
G	General function
H	Tool length offset index
I	X-axis offset for arcs
J	Y-axis offset for arcs
K	Z-axis offset for arcs
L	Number of repetitions in canned cycles and subroutines, or key used with G10
M	Miscellaneous function
N	Line number
O	Subroutine label number
P	Dwell time in canned cycles, dwell time with G04, key used with G10, or tapping depth in M871 through M874
Q	Feed increment in a G83 canned cycle, or repetitions of subroutine call
R	Arc radius, or canned cycle retract level
S	Spindle speed
T	Tool selection
U	Synonymous with A

Letter	Description
V	Synonymous with B
W	Synonymous with C
X	X-axis
Y	Y-axis
Z	Z-axis

Values

A real value is one of the following:

- An explicit number (like 341, or -0.8807)
- An expression (like [2+2.4])
- A parameter value (like #88)
- A unary operation value (like acos[0])



Note: In the command examples that we use, the tilde symbol (~) stands for a real value. If L~ is written in an example, the ~ is often referred to as the L number. Similarly the ~ in H~ may be called the H number, and so on for any other letter.

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:

- Comment (including message)
- Set feed rate mode (`G93`, `G94`, `G95`)
- Set feed rate (`F`)
- Set spindle speed (`S`)
- Special I/O (`M62` to `M68`)



Note: This is not supported.

- Change tool (`T`)
- Spindle on/off (`M03`, `M04`, `M05`)
- Save State (`M70`, `M73`, restore state (`M72`), invalidate state (`M71`)
- Coolant on/off (`M07`, `M08`, `M09`)
- Enable/disable overrides (`M48`, `M49`, `M50`, `M51`, `M52`, `M53`)
- Operator defined commands (`M101` to `M199`)
- Dwell (`G04`)
- Set active plane (`G17`, `G18`, `G19`)
- Set length units (`G20`, `G21`)
- Cutter radius compensation on/off (`G40`, `G41`, `G42`)
- Tool table offset on/off (`G43`, `G49`)
- Fixture table select (`G54` through `G58` and `G59 P~`)
- Set path control mode (`G61`, `G61.1`, `G64`)
- Set distance mode (`G90`, `G91`)
- Set canned cycle return level mode (`G98`, `G99`)

9: PROGRAMMING
9.2 Programming Overview

- 21. Home, change coordinate system data (G10) or set offsets (G92, G94)
- 22. Perform motion (G00 to G03, G12, G13, G80 to G89 as modified by G53)
- 23. Stop (M00, M01, M02, M30, M60)

Modal Groups

G- and M-codes are, generally speaking, modal — they cause the machining system to change from one mode to another. The mode stays active until another command changes it implicitly or explicitly.

EXAMPLE
If coolant is turned on (M07 or M08), it stays on until it is explicitly turned off in the program (M09).

A few G-codes and M-codes are non-modal (like Dwell (G04)). These codes have effect only on the lines on which they occur. Modal commands are arranged in sets, called modal groups. Only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time (like inch units (G20) vs. millimeter units (G21)). A machining system may be in many modes at the same time, with one mode from each modal group being in effect. For all G-code modal groups, when a machining system is ready to accept commands, one member of the modal group must be in effect. There are default settings for these modal groups. When the machining system is turned on or re-initialized, default values are automatically in effect. Modal groups for G-codes are detailed in the following table.

Table with 3 columns: Group, Commands, Group Description. Row 1: 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)

Table with 3 columns: Group, Commands, Group Description. Rows include: Group 2 (Plane selection), Group 3 (Distance mode), Group 4 (Arc distance mode), Group 5 (Feed rate mode), Group 6 (Length units), Group 7 (Cutter compensation), Group 8 (Tool length offset), Group 10 (Return mode in canned cycles), Group 12 (Select work offset coordinate system), Group 13 (Path control mode), Group 14 (Spindle control mode), Group 15 (Lathe diameter mode)

Modal groups for M-codes are detailed in the following table.

Table with 3 columns: Group, Commands, Group Description. Row 1: Group 4, {M00, M01, M02, M30, M60}, Program stop and program end. Row 2: Group 7, {M03, M04, M05}, Spindle control

Group	Commands	Group Description
Group 8	{M07, M08, M09}	Coolant control (special case: M07 and M08 may be active at the same time)
Group 9	{M48, M49}	Override control

Non-modal G-codes are:

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

Comments

You can add comments to lines of G-code to help clarify the intention of the programmer. To embed a comment in a line, use parentheses. To add a comment to the end of a line, use a semicolon.



Note: The semicolon is not treated as the start of a comment when it's enclosed in parentheses.

Comments can appear between words, but they can't be between words and their corresponding parameter.



EXAMPLE :

S100 (set speed) F200 (feed) is okay, but S (speed) 100F (feed) is not.

9.2.3 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
<u>G10 L1</u>	Set tool table
<u>G10 L2</u>	Set coordinate system
<u>G10 L10</u>	Set tool table – calculated – workpiece
<u>G10 L11</u>	Set tool table – calculated – fixture
<u>G10 L20</u>	Set coordinate system

G-Code	Description
<u>G15</u>	Workpiece probe
<u>G16</u>	Pierce
<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
<u>G33</u>	Spindle synchronized motion (like threading)
<u>G33.1</u>	Rigid tapping
<u>G37/G37.1</u>	Automatically measure tool lengths with an ETS
<u>G38.x</u>	Straight probe
<u>G40</u>	Cancel cutter compensation
<u>G41/G42</u>	Cutter compensation (left/right)
<u>G41.1, G42.1</u>	Dynamic cutter compensation
<u>G43</u>	Apply tool length offset
<u>G47</u>	Engrave sequential serial number
<u>G49</u>	Cancel tool length 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
<u>G76</u>	Multi-pass threading cycle
<u>G80</u>	Cancel canned cycles
<u>G81</u>	Drilling cycle
<u>G82</u>	Simple drilling cycle
<u>G83</u>	Peck drilling cycle
<u>G85</u>	Boring cycle

9: PROGRAMMING

9.2 Programming Overview

G-Code	Description
<u>G86</u>	Boring cycle
<u>G88</u>	Boring cycle
<u>G89</u>	Boring cycle
<u>G90</u> , <u>G90.1</u>	Arc distance mode
G91, G91.1	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92, etc.
<u>G93</u> , <u>G94</u> , <u>G95</u>	Feed rate mode
<u>G96</u> , <u>G97</u>	Spindle control mode
G98	Initial level return / R-point level after canned cycles

9.3 PROGRAMMING G-CODE

Read the following sections as a G-code reference:

9.3.1 About the Examples Used	137
9.3.2 Rapid Linear Motion (G00).....	137
9.3.3 Linear Motion at Feed Rate (G01).....	138
9.3.4 Arc at Feed Rate (G02 and G03).....	138
9.3.5 Dwell (G04).....	141
9.3.6 Set Offsets (G10).....	141
9.3.7 Workpiece Probe (G15).....	142
9.3.8 Pierce (G16).....	142
9.3.9 Plane Selection (G17, G18, G19).....	142
9.3.10 Length Units (G20 and G21).....	142
9.3.11 Return to Predefined Position (G28 and G28.1)....	142
9.3.12 Return to Predefined Position (G30 and G30.1)....	142
9.3.13 Automatically Measure Tool Lengths with an ETS (G37 and G37.1).....	143
9.3.14 Straight Probe (G38.x).....	144
9.3.15 Cutter Compensation (G40, G41, G42).....	145
9.3.16 Dynamic Cutter Compensation (G41.1 and G42.1).....	145
9.3.17 Apply Tool Length Offset (G43).....	145
9.3.18 Engrave Sequential Serial Number (G47).....	145
9.3.19 Cancel Tool Length Compensation (G49).....	146
9.3.20 Absolute Coordinates (G53).....	146
9.3.21 Select Work Offset Coordinate System (G54 to G54.1 P500).....	146
9.3.22 Set Exact Path Control Mode (G61).....	147
9.3.23 Set Blended Path Control Mode (G64).....	147
9.3.24 Distance Mode (G90 and G91).....	147
9.3.25 Arc Distance Mode (G90.1 and G91.1).....	147
9.3.26 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....	147
9.3.27 Feed Rate Mode (G93, G94, and G95).....	148
9.3.28 Spindle Control Mode (G96 and G97).....	148

9.3.1 About the Examples Used

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

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

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

9.3.2 Rapid Linear Motion (G00)

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

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

This produces coordinated linear motion to the destination point at the current traverse rate (or slower, if the machine won't go that fast). It's expected that cutting won't take place when a G00 command is executing. The G00 is optional if the current motion mode is G00.

Depending on where the tool is located, follow these two basic rules:

1. If the Z value represents a cutting move in the positive direction (like out of a hole), the X-axis should be moved last.
2. If the Z value represents a move in the negative direction, the X-axis should be moved first.

Conditions

The motion differs if:

- Cutter radius compensation is active
- G53 is programmed on the same line

Troubleshooting

It's an error if:

- All axis words are omitted
The axis words are optional, except that at least one must be used.
- G10, G28, G30 or G92 appear in the same block

9: PROGRAMMING

9.3 Programming G-Code

9.3.3 Linear Motion at Feed Rate (G01)

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

```
G01 X~ Y~ Z~ A~ F~
```

- X~ is the X-axis coordinate
- Y~ is the Y-axis coordinate
- Z~ is the Z-axis coordinate
- A~ is the A-axis coordinate
- F~ is the feed rate

This produces coordinated linear motion to the destination point at the current feed rate (or slower, if the machine won't go that fast). The G01 is optional if the current motion mode is G01.

Conditions

The motion differs if:

- Cutter radius compensation is active
- G53 is programmed on the same line

Troubleshooting

It's an error if:

- All axis words are omitted
The axis words are optional, except that at least one must be used.
- G10, G28, G30, or G92 appear in the same block
- No F word is specified

9.3.4 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 or helix must be parallel to the X-, Y- or Z-axis of the machine coordinate system. The axis (or equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane) or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane. If a line of code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. This is rare.

The motion differs if cutter radius compensation is active. Two formats are allowed for specifying an arc: the center format and the radius format. In both formats, the G02 or G03 is optional if it's the current motion mode.

Radius Format Arc

For a clockwise arc in radius format, program: G02 X~ Y~

```
Z~ A~ R~
```

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

```
Y~ Z~ A~ R~
```

- X~ is the X-axis coordinate
- Y~ is the Y-axis coordinate
- Z~ is the Z-axis coordinate
- A~ is the A-axis coordinate
- R~ is the radius of the arc

In radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees.

If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified. We don't recommend programming radius format arcs that are:

- Nearly full circles
- Semicircles
- Nearly semicircles

A small change in the location of the end point produces a much larger change in the location of the center of the circle (and the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts.

You can program arcs that are:

- Up to 165 degrees
- Between 195 degrees to 345 degrees



EXAMPLE

G17 G02 X 1.0 Y 1.5 R 2.0 Z 0.5 is a radius format command to mill an arc, which makes a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X = 1.0, Y = 1.5, and Z = 0.5, with a radius of 2.0. If the starting value of Z is 0.5, this is an arc of a circle parallel to the XY-plane; otherwise, it's a helical arc.

Troubleshooting

It's an error if:

- Both of the axis words for the axes of the selected plane are omitted
The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used.
- No R word is given
- The end point of the arc is the same as the current point
- G10, G28, G30, or G92 appear in the same block

Center Format Arc

For a clockwise arc in center format, program: G02 X~ Y~

Z~ I~ J~

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

Y~ Z~ I~ J~

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

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

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

It's an error if:

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

Arc in XY Plane

When the XY-plane is selected, program: G02 X~ Y~ Z~ A~

I~ J~ (or, use G03 instead of G02)

I and J are the offsets from the current location or coordinates – depending on arc distance mode (G90.1/G91.1) of the center of the circle (X and Y directions, respectively).

It's an error if:

- X and Y are both omitted
The axis words are all optional except that at least one of X and Y must be used.
- I and J are both omitted
I and J are optional except that at least one of the two must be used.

Arc in XZ Plane

When the XZ-plane is selected, program: G02 X~ Y~ Z~ A~

I~ K~ (or, use G03 instead of G02)

I and K are the offsets from the current location or coordinates – depending on arc distance mode (G90.1/G91.1) of the center of the circle (X and Z directions, respectively).

It's an error if:

- X and Z are both omitted
The axis words are all optional except that at least one of X and Z must be used.
- I and K are both omitted
I and K are optional except that at least one of the two must be used.

Arc in YZ Plane

When the YZ-plane is selected, program: G02 X~ Y~ Z~ A~

J~ K~ (or, use G03 instead of G02)

J and K are the offsets from the current location or coordinates – depending on arc distance mode (G90.1/G91.1) of the center of the circle (Y and Z directions, respectively).

It's an error if:

- Y and Z are both omitted
The axis words are all optional except that at least one of Y and Z must be used.
- J and K are both omitted
J and K are optional except that at least one of the two must be used.

9: PROGRAMMING

9.3 Programming G-Code



EXAMPLE

G17 G02 X1.0 Y1.6 I0.3 J0.4 Z0.9 is a center format command to mill an arc in incremental arc distance mode (G91.1) that makes a clockwise (as viewed from the positive Z-axis), circular, or helical arc whose axis is parallel to the Z-axis, ending where X = 1.0, Y = 1.6, and Z = 0.9, with its center offset in the X direction by 0.3 units from the current X location and offset in the Y direction by 0.4 units from the current Y location. If the current location has X = 0.7, Y = 0.7 at the outset, the center is at X = 1.0, Y = 1.1. If the starting value of Z is 0.9, this is a circular arc; otherwise, it's a helical arc. The radius of this arc would be 0.5.

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

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

9.3.5 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

9.3.6 Set Offsets (G10)

Use the controls on the Offsets tab to set offsets. You can program offsets with the G10 G-code command.

Read the following sections for reference:

Set Tool Table (G10 L1).....	141
Set Tool Table (G10 L10).....	141
Set Tool Table (G10 L11).....	141
Set Coordinate System (G10 L20).....	141

Set Tool Table (G10 L1)

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

- P~ is the tool number
- R~ is the radius of tool

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

Troubleshooting

It's an error if:

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

Set Tool Table (G10 L10)

To change the tool table entry for tool P so that if the tool offset is reloaded with the machine in its current position and with the current G5x and G92 offsets active, program: G10

L10 P~ R~

- P~ is the tool number
- R~ is the radius of tool

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

Troubleshooting

It's an error if:

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

Set Tool Table (G10 L11)

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

Program: G10 L11 P~ X~ Y~ Z~ R~

- P~ is the tool number
- R~ is the radius of tool

Troubleshooting

It's an error if:

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

Set Coordinate System (G10 L20)

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

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

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

9: PROGRAMMING

9.3 Programming G-Code

- $Y\sim$ is the Y-axis coordinate
- $Z\sim$ is the Z-axis coordinate
- $A\sim$ is the A-axis coordinate

Troubleshooting

It's an error if:

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

9.3.7 Workpiece Probe (G15)

Probe vertically downwards towards the end of travel until the ohmic cap contacts the work piece.

Once the workpiece has been located, the Z coordinate of the current coordinate system is set to zero.

G15 will trigger on either the ohmic cap making continuity with the workpiece or the physical touch switch in the torch lifter detecting upwards pressure on the torch from the material.

9.3.8 Pierce (G16)

Raise the torch to the pierce height set in the variable #<_PierceHeight>. The torch is turned on and the program pauses for the pierce delay value set by AutoFS or M209. After the pierce delay, Torch Height Control (THC) is enabled and the program continues.

9.3.9 Plane Selection (G17, G18, G19)

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

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

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

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

9.3.10 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.

9.3.11 Return to Predefined Position (G28 and G28.1)

To make a rapid linear move from the current position to the absolute position of the values in parameters 5161-5166: G28

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



Note: Any axis not specified won't move.

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

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

To store the current absolute position into parameters 5161-5163, program: G28.1

Troubleshooting

It's an error if:

- Cutter Compensation is turned on

9.3.12 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 $X\sim$ $Z\sim$



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

9.3.13 Automatically Measure Tool Lengths with an ETS (G37 and G37.1)

Use G37 and G37.1 with an Electronic Tool Setter (ETS) to enable automatic length measurement. For automated use, add a G37 command after an M6 tool change commands. If you're using the ETS with a mill, the input port varies depending on your machine:

- **M Series Mills** Plug the ETS into the Accessory Input 2 port. You can still use the Accessory Input 1 port for other probes and accessories.
- **Older PCNC Mills** Plug the ETS into the (single) accessory input port.

Move to G37 Position Over ETS (G37.1)

To move to the G37 position (over the ETS), program: G37.1
To set the G37 position:

1. Jog the machine over the center of the ETS.
2. From the **Probe** tab, on the **ETS Setup** tab, select **Set G37 ETS Position**.

The read-only DROs in the **ETS G37 Position Setup** group display the new position.

The G37 position is in G53 machine coordinate space. It defaults to (0, 0, 0), or the top left rear of machine travel (the same as the X-, Y-, and Z-axis reference position).

G37.1 supports X and Y tool offsets. If there are X or Y tool offsets present in the tool table (manually applied through a G10 L1 command), they offset the spindle position. This enables G37 for tools mounted in an auxiliary spindle installed on the spindle column.



Note: If G10 L1 is used to change the X or Y offset of the currently loaded tool, you must then apply the new offsets with a G43 command.

G37.1 performs as follows:

1. A rapid upward move to the Z clear position (which is always G53 Z = 0.0).
2. A rapid move in X and Y to the X and Y ETS coordinates.
3. A rapid downward move in Z to the Z ETS coordinate.



Note: The Z word saves time by rapidly moving closer to the ETS before the slower probing begins. You must use caution if you set this lower than G53 Z = 0.0. If you don't, there's a risk that a long tool could collide with the ETS and damage it and the tool.

Move and Measure Tool Length (G37)

To move and measure the tool length, program: G37 H~ P~

- **H~** saves the measured tool length to the H tool table entry instead of the current tool number's entry. You could use this to track tool wear between the two tool table entries, for example.



Note: The newly measured tool length isn't applied, but it's stored in the tool table entry for tool number H.

- **P~** is positive or negative tolerance. It measures the tool length, but, instead of storing it in the tool table, compares it to the length in the tool table. If the difference exceeds the P tolerance, the G-code program stops. You could use this to detect broken or improperly inserted tools that are not fully seated in the spindle, for example.

G37 with no optional words moves to the G37 ETS position (through G37.1), probes the ETS, stores the new tool length in the tool table entry of the current tool, and applies the tool length offset.

G37 fails if the spindle nose hasn't been referenced to the ETS after a Z-axis reference. This sets a G53 coordinate at the ETS trigger point such that the measured tool length is the distance of the spindle nose to ETS reference. For more information, see the **ETS G37 Spindle Nose Reference** group on the **ETS Setup** tab.

So that tool length measurements have consistent results, G37 uses the fine probe feed rate of 2.5 in./min for the final ETS touch. G37 uses the rough probe feed rate for the first ETS touch.

G37 performs as follows:

1. Issues a G37.1 move to the ETS location.
2. A downward rough probe feed rate move until the tool triggers the ETS.

9: PROGRAMMING

9.3 Programming G-Code

3. An upward retract move of 0.100 in. to back off the triggered ETS.
4. A downward slow ETS probe feed rate move until the tool triggers the ETS.
5. An upward retract move of 0.100 in. to back off the triggered ETS.
6. An upward rapid move to the G37 ETS Z position.

9.3.14 Straight Probe (G38.x)

G38.2 probes toward the workpiece, stops on contact, and signals error if failure

G38.3 probes toward the workpiece and stops on contact

G38.4 probes away from the workpiece, stops on loss of contact, and signals error if failure

G38.5 probes away from the workpiece and stops on loss of contact

G38.6 moves away from the workpiece and ignores probe input

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

Conventionally, the probe is tool #99. The rotational axis words are allowed, but it's better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The tool in the spindle must be a probe.

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

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

Troubleshooting

It's an error if:

- The current point is less than 0.01 in. (0.254 mm) from the programmed point
 - G38 is used in inverse time feed rate mode
 - Any rotational axis is commanded to move
 - No X-, Y- or Z-axis word is used
- The linear axis words are optional, except that at least one of them must be used.

- Feed rate is zero
- The probe is already tripped

Use the Straight Probe Command

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

- Without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane
- If the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane
- If the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known approximately, the center of a circular hole

If the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, you can use the straight probe command for things like finding the diameter of a circular hole.

Example code:

```
o<probe_pocket> sub
(probe to find center of circular or rectangular
pocket)
#<x_start> = #5420 (Current X Location)
#<y_start> = #5421 (Current Y Location)
#<x_max> = 1
#<x_min> = -1
#<y_max> = 1
#<y_min> = -1
#<feed_rate> = 30 (30 IPM)
F #<feed_rate>
G38.3 X #<x_max> (rough probe +X side of hole)
F [#<feed_rate>/30]
G38.5 X #<x_start> (finish probe)
#<x_plus>=#5061 (save results)
G00 X #<x_start> (return to start)
F #<feed_rate>
G38.3 X #<x_min> (probe -X side of hole)
F [#<feed_rate>/30]
G38.5 X #<x_start>
#<x_minus>=#5061 (save results)
G00 X #<x_start>
#<x_center> = [(#<x_plus>+#<x_minus>)]/2]
G00 X #<x_center> (go to middle)
```

```
F #<feed_rate>
G38.3 Y #<y_max> (probe +Y side of hole)
F [#<feed_rate>/30]
G38.5 Y #<y_start>
#<y_plus>=#5062 (save results)
G00 Y #<y_start> (return to start)
F #<feed_rate>
G38.3 Y #<y_min> (probe -Y side of hole)
F [#<feed_rate>/30]
G38.5 Y #<y_start>
#<y_minus>=#5062 (save results)
G00 Y #<y_start>
#<y_center> = [(#<y_plus>+#<y_minus>)/2]
G00 Y #<y_center> (go to middle)
G10 L20 P1 X 0 Y 0 (set current location to zero)
F #<feed_rate> (restore original feed rate)
o<probe_pocket> endsub
M02
```

9.3.15 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

9.3.16 Dynamic Cutter Compensation (G41.1 and G42.1)

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

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

- D~ is the tip radius multiplied by two

G41.1 and G42.1 function the same as G41 and G42, with the added scope of being able to ignore the tool table and to program the tool diameter.

Troubleshooting

It's an error if:

- Cutter Compensation is commanded to turn on when it is already on

9.3.17 Apply Tool Length Offset (G43)

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

- H~ is the tool number associated with the length offset to be applied.



Note: Generally speaking, the value of the H~ word should match the active tool number (T~ word).

It's okay to program using the same offset already in use, or to program without a tool length offset (if none is currently being used).

Troubleshooting

It's an error if:

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

9.3.18 Engrave Sequential Serial Number (G47)

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

- Z~ is the depth of cut of the engraving
- R~ is the retract height between character segments in the numbers
- X~ is, if present, the starting X position, or the left side of the serial number
If omitted, the current X position is assumed.

9: PROGRAMMING

9.3 Programming G-Code

- **Y~** is, if present, the starting Y position, or the bottom side of the serial number
If omitted, the current Y position is assumed.
- **P~** is, if present, the X extent (width) in current units (inches or millimeters) of the engraved number
- **Q~** is, if present, the Y extent (height) in current units (inches or millimeters) of the engraved number
- **D~** is, if present, the requested number of decimals of the engraved number
If the requested D value exceeds the number of decimals in the serial number, the serial number will show leading zeros. If the requested D value is less than the number of decimals in the serial number, only the digits of the serial number will show.



EXAMPLE

A serial number of 10, where D = 4, engraves as 0010. A serial number of 9056, where D = 3, engraves as 9056.

Troubleshooting

It's an error if:

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

9.3.19 Cancel Tool Length Compensation (G49)

To cancel tool length compensation, program: **G49**

9.3.20 Absolute Coordinates (G53)

For rapid linear motion to a point expressed in absolute coordinates, program: **G01 G53 X~ Y~ Z~** (or use with **G00** instead of **G01**)

All the axis words are optional, except that at least one must be used. The **G00** or **G01** is optional if it is in the current motion mode. **G53** isn't modal, and must be programmed on each line on which it is intended to be active. This produces coordinated linear motion to the programmed point. If **G01** is active, the speed of motion is the current feed rate (or slower if the machine won't go that fast). If **G00** is active, the speed of motion is the current traverse rate (or slower if the machine won't go that fast).

Troubleshooting

It's an error if:

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

9.3.21 Select Work Offset Coordinate System (G54 to G54.1 P500)

You can save up to 500 work offsets in PathPilot. The naming structure varies based on the offset number.

- To select work offset 1, program: **G54** or **G54.1 P1**
- To select work offset 2, program: **G55** or **G54.1 P2**
- To select work offset 3, program: **G56** or **G54.1 P3**
- To select work offset 4, program: **G57** or **G54.1 P4**
- To select work offset 5, program: **G58** or **G54.1 P5**
- To select work offset 6, program: **G59** or **G54.1 P6**
- To select work offset 7, program: **G59.1** or **G54.1 P7**
- To select work offset 8, program: **G59.2** or **G54.1 P8**
- To select work offset 9, program: **G59.3** or **G54.1 P9**
- To select a work offset beyond the standard 9 (listed above), program: **G54.1 P###**, where **P###** is a parameter indicating the index of the work offset you want to use (work offset 10 through work offset 500).



EXAMPLE

To select the 124th work offset, program **G54.1 P124**.

For information, see "About Work Offsets" (page 105).

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).

9.3.22 Set Exact Path Control Mode (G61)

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

9.3.23 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.

9.3.24 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.

9.3.25 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.

9.3.26 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).

9: PROGRAMMING

9.3 Programming G-Code

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`

9.3.27 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`

9.3.28 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

9.4 PROGRAMMING M-CODE

Read the following sections for reference:

9.4.1 Supported M-Codes Reference.....	149
9.4.2 Program Stop and Program End (M00, M01, M02, and M30).....	149
9.4.3 Spindle Control (M03, M04, and M05).....	150
9.4.4 Tool Change (M06).....	150
9.4.5 Coolant Control (M07, M08, and M09).....	151
9.4.6 Override Control (M48 and M49).....	151
9.4.7 Feed Override Control (M50).....	151
9.4.8 Spindle Speed Override Control (M51).....	151
9.4.9 Set Current Tool Number (M61).....	151
9.4.10 Set Output State (M64 and M65).....	151
9.4.11 Wait on Input (M66).....	152
9.4.12 USB Camera Control (M301, M302, M303).....	152
9.4.13 Plasma Specific M-Codes.....	152

9.4.1 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>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>M66</u>	Wait on an input



Note: M64 through M66 is only useful with a [USB M-Code I/O Interface Kit \(PN 32616\)](#).

M-Code	Description
M98	Call subroutine
M99	Return from subroutine/repeat
<u>M200</u> , <u>M203</u> , <u>M205</u> , <u>M207</u> - <u>M13</u>	Plasma specific M-codes
<u>M301</u> , <u>M302</u> , <u>M303</u>	USB camera control

9.4.2 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)

9: PROGRAMMING

9.4 Programming M-Code

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

Display Information and Capture Images During an M00 or M01 Break

Display Information with Images

If the comment occurs on a line with **M00** or **M01**, and contains a file name with a .jpg or .png extension, PathPilot displays the 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 101).

9.4.3 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.

9.4.4 Tool Change (M06)

To execute a tool change sequence, program: **M06**

M06 behaves differently depending on whether or not the machine is equipped with an Automatic Tool Changer (ATC):

- If you have an ATC:
 - If the requested tool (T number) is assigned to the carousel, **M06** initiates an automatic tool change.
 - If the tool is not assigned to the carousel, you're prompted to manually change the tool and select Cycle Start to confirm the tool change. This resumes the program.

- If you don't have an ATC:
 - M06 commands the machine, stops the spindle, pauses program execution, and prompts operator to change tools by flashing Tool Change LED.
 - The program resumes after you select Cycle Start to confirm that the tool has been changed.

We recommend putting the T~, the M06, and the G43 H~ on one line (block) of code.

 **EXAMPLE**
N191 M06 T3 G43 H3

9.4.5 Coolant Control (M07, M08, and M09)

To turn coolant on, program: M07

To turn flood coolant on, program: M08

To turn all coolant off, program: M09

It's always okay to use any of these commands, regardless of what coolant is on or off.

9.4.6 Override Control (M48 and M49)

To enable the speed and feed override, program: M48

To disable both overrides, program: M49

It's okay to enable or disable the switches when they are already enabled or disabled.

9.4.7 Feed Override Control (M50)

To enable the feed rate override control, program: M50 P1

The P1 is optional.

To disable the feed rate control, program: M50 P0

When feed rate override control is disabled, the feed rate override slider has no influence, and all motion is executed at programmed feed rate (unless there is an adaptive feed rate override active).

9.4.8 Spindle Speed Override Control (M51)

To enable the spindle speed override control, program: M51 P1

The P1 is optional.

To disable the spindle speed override control, program: M51 P0

When spindle speed override control is disabled, the spindle speed override slider has no influence, and the spindle speed is equal to the value of the S word.

9.4.9 Set Current Tool Number (M61)

To change the current tool number while in MDI or manual mode, program: M61 Q~

- Q~ is the tool number

Troubleshooting

It's an error if:

- Q~ is not 0 or greater

9.4.10 Set Output State (M64 and M65)

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

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

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

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

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

 **EXAMPLE**

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

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

 **EXAMPLE**

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

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

The following is legal:

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

The following is not legal:

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

9: PROGRAMMING

9.4 Programming M-Code

9.4.11 Wait on Input (M66)



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

There are four digital inputs available on the USB I/O module.

M66 P- | E- <L->

- P- is the digital input number from 0 to 3.
- L- is the wait mode type:
 - Mode 0: IMMEDIATE – no waiting, returns immediately. The value of the input at that time is stored in parameter #5399.
 - Mode 1: RISE – waits for the selected input to perform a rise event.
 - Mode 2: FALL – waits for the selected input to perform a fall event.
 - Mode 3: HIGH – waits for the selected input to go to the HIGH state.
 - Mode 4: LOW – waits for the selected input to go to the LOW state.
- Q- is the timeout in seconds for waiting

The Q value is ignored if the L word is zero (IMMEDIATE). A Q value of zero is an error if the L word is non-zero.

9.4.12 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 101).

9.4.13 Plasma Specific M-Codes

M-Code	Description	Set by M200	Parameters
M200	AutoFS - Apply the automatically generated feed, speed and cutting values for the material selected in the main PathPilot UI. AutoFS will set feedrate, THC voltage, and pierce delay.		
M203	Start Torch		
M205	Stop Torch and return to G30 Z Height		
M207	Set Pierce Height used when executing a G16 pierce. Set automatically when using M200 AutoFS.	Yes	P (Required): Pierce height in the current unit system (in. or mm.)
M208	Set Cut Height used when executing a G16 pierce. Set automatically when using M200 AutoFS.	Yes	P (Required): Cut height in the current unit system (in. or mm.)
M209	Set Pierce Delay used when executing a G16 pierce. Set automatically when using M200 AutoFS.	Yes	P (Required): Pierce delay in seconds
M210	Set THC voltage. Does not enable or disable THC.	Yes	P (Required): Desired THC Voltage

M-Code	Description	Set by M200	Parameters
M211	Set plasma source cut current. If a Hypertherm adapter is not installed on the system, program run will be paused with an M01 break to allow the user to set the specified cut current.	Yes	P (Required): Desired Cut Current in Amps
M212	Set plasma source cut mode. If a Hypertherm adapter is not installed on the system, program run will be paused with an M01 break to allow the user to set the specified cut mode.	No	P (Required): Cut mode: Normal = 1 CPA = 2 Gouge = 3 Note: For the Hypertherm Powermax 45 Normal Mode and CPA Mode (Continuous Pilot Arc) are the same.
M213	Set plasma source air pressure. If a Hypertherm adapter is not installed on the system, program run will be paused with an M01 break to allow the user to set the specified air pressure.	No	P: Desired Cut Pressure in PSI Note: When the pressure is omitted or set to 0, the pressure control is set to Automatic.

9: PROGRAMMING

9.5 Programming Input Codes

9.5 PROGRAMMING INPUT CODES

Read the following sections for reference:

9.5.1 Feed Rate (F)	154
9.5.2 Spindle Speed (S)	154
9.5.3 Change Tool Number (T)	154

9.5.1 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.

9.5.2 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

9.5.3 Change Tool Number (T)

It's your responsibility to make sure that the machine is in a safe place for changing tools (for example, by using `G30`). This allows optimization of motion which can save time. You can provide a pause for manual intervention with `M00` or `M01` before the tool change.

Troubleshooting

It's an error if:

- A negative T number is used
- A T number larger than 1000 is used

9.6 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:

9.6.1 Parameters	155
9.6.2 Expressions	157
9.6.3 Subroutines	158

9.6.1 Parameters

Read the following sections for reference:

Parameters Reference	155
Numbered Parameters Reference	156
Subroutine Parameters Reference	156
Named Parameters Reference	156

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

9: PROGRAMMING

9.6 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
- 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.

9.6.2 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.....	157
Functions Reference.....	157

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)

9: PROGRAMMING
9.6 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

9.6.3 Subroutines

Subroutines are subprograms that are called from inside another program.
Read the following sections for reference:

Subroutines Reference.....158
Conditional Subroutines Reference..... 159
Repeating Subroutines Reference..... 159
Looping Subroutines Reference.....159

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.

9: PROGRAMMING

9.6 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
```

while/endwhile

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

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

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

- A return or endsub not within a sub definition
- A label on repeat which is defined elsewhere
- A label on while which is defined elsewhere and not referring to a do
- A label on if defined elsewhere
- A undefined label on else or elseif
- A label on else, elseif or endif not pointing to a matching if
- A label on break or continue which does not point to a matching while or do
- A label on endrepeat or endwhile no referring to a corresponding while or repeat

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

10.1 Maintenance Safety	162
10.2 Maintenance Schedules	163

10: MACHINE MAINTENANCE

10.1 Maintenance Safety

10.1 MAINTENANCE SAFETY

Read and understand the following safety messages before beginning any maintenance procedures.

10.1.1 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 and the pneumatic supply line before doing any maintenance procedures.
- ✓ Wear safety eye protection rated for ANSI Z87+.

10.1.2 Swarf Maintenance Procedures

- ✓ Wear work gloves.

10.2 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 179). 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.

10.2.1 Daily

- ☐ Clean the machine of any dust or abrasive particle buildup with a vacuum and brushes.
- ☐ Wipe dust off of the linear rails with a clean cloth.

10.2.2 Weekly

- ☐ Clean all exterior surfaces with a clean rag.
- ☐ Examine the water table's level and, if necessary, add water / rust inhibitor.
- ☐ Check your plasma torch consumables for wear and replace if needed.

10.2.3 Monthly

- ☐ Clean the electrical cabinet vents of dust with a clean cloth or compressed air.
- ☐ Remove the slats and clean sediment buildup from the bottom of the water table.

10.2.4 Semi-Annually

- ☐ Check drive belts for wear and re-tension if needed.
- ☐ Remove faceplate from torch lifter and clean lead screw with a lint-free cloth.

10.2.5 As Needed

- ☐ Replace the water table's slats when they have been worn away too much to hold stock securely.

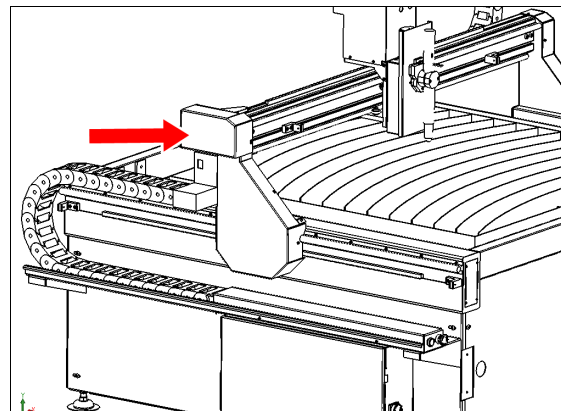
10.2.6 Adjust Belt Tension

Tools and Items Required

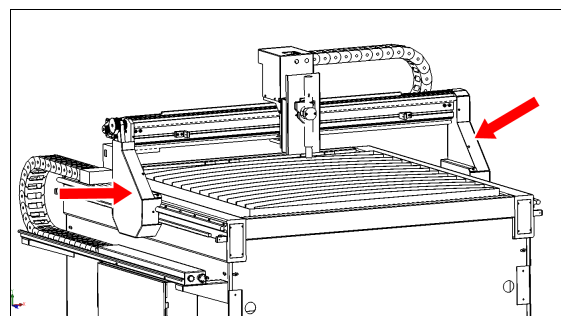
- Allen Key Set

The drive belts on the 1300PL should be checked for wear and tensioned several times a year to avoid lost motion and reduced machine accuracy.

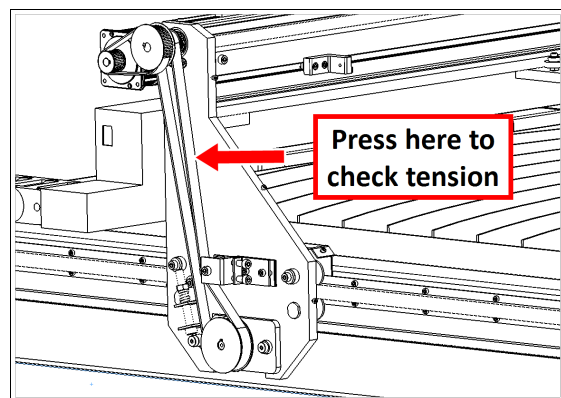
1. Remove the motor belt cover over the servo motor pulley.



2. Remove the belt covers from both sides of the gantry.



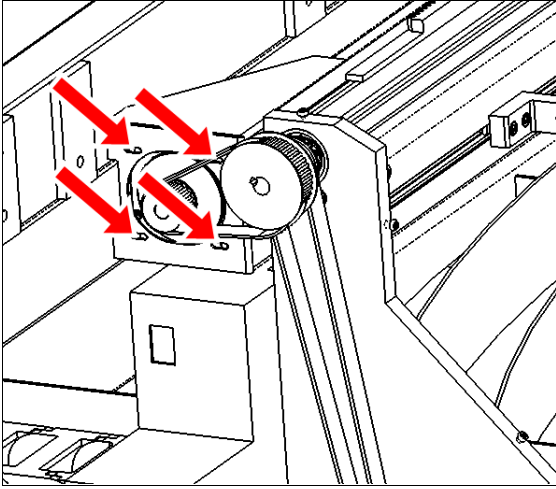
3. Loosen the two mounting bolts holding the belt tensioner. Adjust the tensioner until the belt is tight enough to only deflect 0.125" (3 mm) when pressed with one finger.



10: MACHINE MAINTENANCE

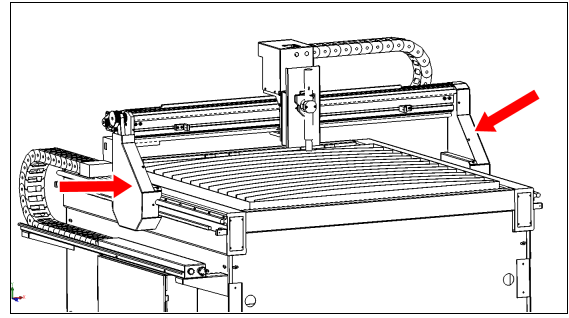
10.2 Maintenance Schedules

4. Tighten the tensioner mounting hardware and repeat for the other side of the gantry.
5. Loosen the mounting hardware for the servo motor.

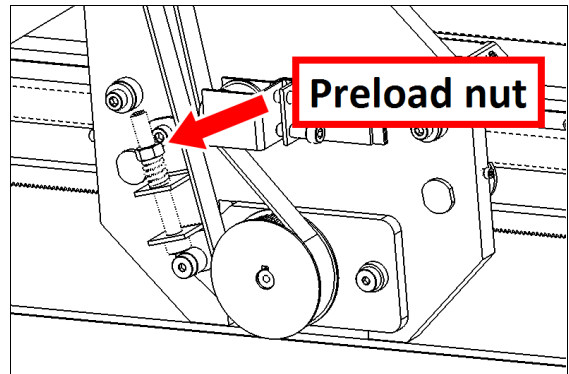


6. Slide the servo back in its mounting slots until the belt is taught. It should deflect about 1/16" (1.5 mm) when pressed with one finger.
7. Tighten all mounting hardware and re-install belt covers.

2. Remove the belt covers from both sides of the gantry.



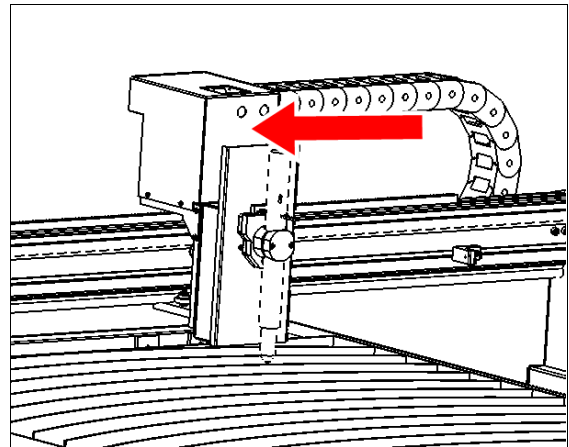
3. Adjust the preload nut on the pinion mount assembly until the spring is compressed approximately 1/4" (6 mm) from its fully slack position.



4. Reinstall all belt covers and guards.

X-Axis

1. Remove the motor belt cover over the servo motor pulley.



10.2.7 Adjust Rack and Pinion Preload

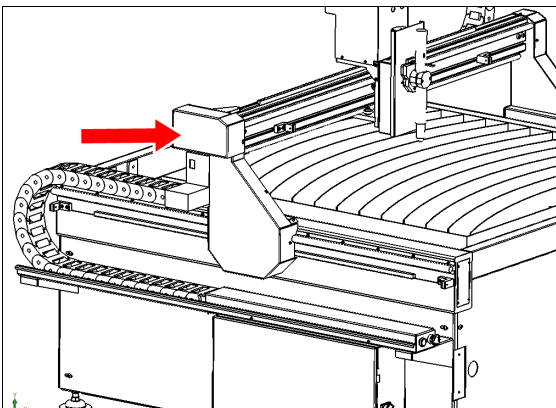
Tools and Items Required

- Allen Key Set
- Crescent Wrench

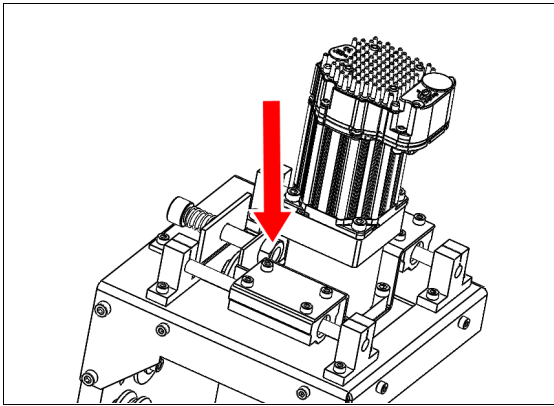
The rack and pinion gear drive on the 1300PL uses a spring preload to allow the pinion gear to self adjust to compensate wear on the rack and reduce backlash. It is preset at the factory and will not need frequent adjustment.

Y-Axis

1. Remove the motor belt cover over the servo motor pulley.



2. Locate and tighten the preload nut for the X motor assembly until the spring is compressed approximately 1/4" (6 mm) from its fully slack position.



3. Reinstall all belt covers and guards.

TORMACH



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

11.1 Ohmic Probing	168
11.2 Touch Off	171
11.3 Cutting	172
11.4 Voltage Feedback	173
11.5 Torch Height Control	175
11.6 Servo Drives	177



TORMACH®

11: TROUBLESHOOTING

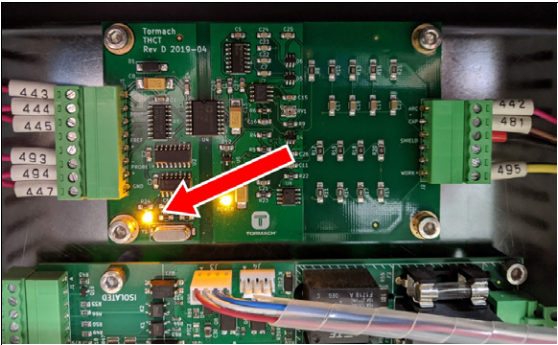
11.1 Ohmic Probing

11.1 OHMIC PROBING

11.1.1 Ohmic Probe Does Not Trigger

How to test ohmic probing:

Open the electrical cabinet and locate the Tormach THCT Board (PN 39096). Have a helper touch the ground clamp to the torch cap and watch LED D8 on the bottom left of the board. If the LED lights up when the ground clamp contacts the torch cap, the THCT board is measuring contact between the cap and ground as intended.



Switch to the Status tab in PathPilot and repeat the test with the ground clamp. Watch the status LED for ohmic touch. If it turns green when the ground clamp makes contact, the THCT board is communicating with PathPilot correctly.



Cause: Faulty Connection Between Workpiece Clamp and Plasma Source

Probability	How-To Steps	Need More?
High	➤ Check that your ground clamp is connected to your workpiece and plasma source.	Ohmic probing detects continuity between the torch cap and the ground connection of the work piece. If this connection is not solid, ohmic probing will not trigger.

Cause: Faulty Work Clamp Connection Between Plasma Source and THCT Board

Probability	How-To Steps	Need More?
Low	1. Open the electrical cabinet and locate the Tormach THCT board (PN 39096). 2. Using a multimeter in continuity mode, measure continuity between J2 Pin 1 (WORK) on the Tormach THCT Board (PN 39096) and the workpiece clamp. 3. If there is not continuity, check that the plasma source control cable is connected at the rear of the electrical cabinet. Pin 4 on this connector is used for the workpiece clamp signal.	Continuity to the workpiece clamp is measured through the plasma source control cable Arc+ pin (since the torch operates at a negative voltage relative to the workpiece). If this connection is faulty or high resistance the probe will not trigger.

Cause: Ohmic Probe Wire Disconnected From the Torch Cap

Probability	How-To Steps	Need More?
Medium	1. Check that the ohmic probe wire (481 on the electrical diagram) coming from the X axis carriage is connected to the torch cap. 2. Check that the connector on the rear of the electrical cabinet (X27 on the electrical diagram) is connected.	To test the wire for breaks, use a multimeter in continuity mode to measure between the torch cap and Pin 5 on connector J2 (labeled CAP) on the Tormach THCT board in the electrical cabinet.

Cause: Ohmic Touch Sensitivity Adjusted Incorrectly

Probability	How-To Steps	Need More?
Low	1. Open the electrical cabinet and locate the Tormach THCT board (PN 39096). 2. Adjust the "RV1" potentiometer clockwise to increase the sensitivity, until a ground clamp touched to the torch nose triggers the probe.	Sensitivity is adjustable to account for different workpiece materials and situations (wet metal). See "Calibrate Initial Height Sensing" (page 62).

Cause: THCT Board Disconnected From the Machine Control Board

Probability	How-To Steps	Need More?
Low	1. Open the electrical cabinet and locate the Tormach THCT board (PN 39096). 2. Using a multimeter on continuity mode, check the connection between THCT J1 Pin 6 (PROBE-) and J9 Pin 5 on the main control board. 3. Perform the same check between THCT J1 Pin 7 (PROBE+) and J9 Pin 6 on the main control board. 4. If both connections measured good, switch the multimeter to DC Voltage mode and measure the voltage between J9 Pin 6 and 7 on the main control board. The reading should be -3.5v without the probe tripped, and +3.5v with the probe tripped.	

11.1.2 Ohmic Probe Triggers Early or Between Cuts**Cause: Plasma Torch Cap Is Wet**

Probability	How-To Steps	Need More?
High	1. Use the TEST TORCH button to purge the torch cap. Make sure to jog away from the workpiece first. 2. Using compressed air, blow any water out of your torch cap. 3. Adjust ohmic touch sensitivity as detailed below.	Water in the torch cap can cause ohmic probing to trigger because it creates a circuit between the torch nozzle (Arc-) and cap. Since the nozzle is electrically connected to the workpiece clamp (Arc+) by the arc voltage divider, this can trigger the ohmic probe.

11: TROUBLESHOOTING

11.1 Ohmic Probing

Cause: Ohmic Touch Sensitivity Adjusted Incorrectly

Probability	How-To Steps	Need More?
High	1. Position the torch in above the workpiece in a small puddle of water, so that the torch cap is touching the water but not the workpiece. 2. Open the electrical cabinet and locate the Tormach THCT board (PN 39096). 3. Adjust the "RV1" potentiometer counterclockwise to decrease the sensitivity until the ohmic probe is not triggered by the water but is still triggered if you lift the workpiece to touch the torch cap.	Ohmic touch sensitivity may need to be adjusted when you switch between styles of consumable or torch cap.

11.2 TOUCH OFF

11.2.1 The Torch Pierces Too High

Cause: Torch Touch Trigger Depth Set Incorrectly		
Probability	How-To Steps	Need More?
High	➤ Open the Settings tab and set the Torch Touch Trigger Depth as described in "Plasma Settings Reference" (page 122).	Torch touch trigger depth is the difference between the Z location where the torch touches the material and the Z location where the touch switch in the floating Z head registers a touch-off.
Cause: M207 Pierce Height Set Incorrectly		
Probability	How-To Steps	Need More?
High	➤ If using M200 AutoFS, ensure that you have selected the correct material on the Main tab of PathPilot.	Using the M200 AutoFS system chooses a pierce height appropriate for the material selected on the Main tab.
Medium	➤ If not using M200 AutoFS to set pierce height automatically, make sure that you have a call to M207 in your program to explicitly set a pierce height.	Check "Plasma Specific M-Codes" (page 152) for more information.

11.2.2 The Torch Bends the Workpiece During Touch-off

Cause: Ohmic Probing Disabled		
Probability	How-To Steps	Need More?
High	➤ Open the Settings tab and ensure that the check-box for ohmic probing is enabled.	If ohmic probing is disabled the plasma table will rely on the switch in the floating torch head to detect touch-off. This can be useful for painted materials, but you need to make sure that Torch Touch Trigger Depth is set correctly as noted above. If necessary to work around a bending workpiece, the workpiece or work coordinate system can be relocated to a position where the workpiece is better supported by the table's slats.

11: TROUBLESHOOTING

11.3 Cutting

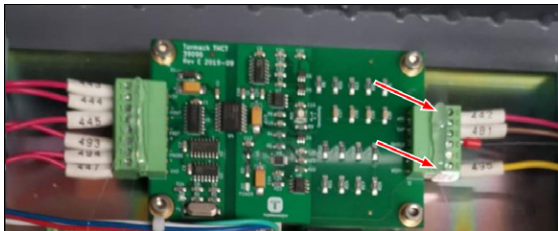
11.3 CUTTING

11.3.1 Torch Shuts Off During Cut

Cause: Air source cannot keep up with demand		
Probability	How-To Steps	Need More?
High	➤ Check the specifications for your compressor to make sure that it can supply the CFM needed by your plasma source. ➤ Check for restrictive fittings or hoses on the air path to your plasma source.	The Hypertherm plasma sources can detect inadequate air flow and will shut off the plasma arc if not enough air is being provided to the unit.
Cause: Gaps in material being cut		
Probability	How-To Steps	Need More?
High	➤ If you are trying to cut across gaps or expanded metal, make sure your plasma source is in "expanded metal mode" and disable Torch Height Control in the PathPilot Settings menu.	Most plasma sources will automatically shut off the arc when they encounter a gap while cutting unless placed in expanded metal / continuous pilot arc mode.

11.4 VOLTAGE FEEDBACK

11.4.1 No Voltage Reading in PathPilot

Cause: Bad control connection to plasma source		
Probability	How-To Steps	Need More?
High	Measure voltage input to THCT board in electrical cabinet: - Using a multimeter set to DC volts, probe wires 442 and 495 on the THCT board with the torch off.  You should read close to 0 VDC. - Have a helper press the TEST TORCH button in PathPilot and check the voltage. You should read your plasma source's open circuit voltage divided by 50 or around 2.5 VDC. - If you do not read a voltage, you may need to replace your plasma source control cable. If you do read a voltage, proceed to checking THCT communication with the Machine Control Board.	Your plasma source is required to have an internal voltage divider. By default, PathPilot is configured for a 50:1 divider.

11: TROUBLESHOOTING

11.4 Voltage Feedback

Cause: THCT not communicating with Machine Control Board

Probability	How-To Steps	Need More?
Low	1. Verify that the THCT board power LED is on  2. Using a multimeter that can measure frequency, measure the signal between wires 444 and 445. You should read a value between 80 kHz and 120 kHz. • If you don't read any frequency, you may have a faulty THCT board. Contact Tormach Technical Support. 3. Have a helper press the TEST TORCH button in PathPilot and check the frequency. You should read a frequency above 200 kHz (usually about 450 kHz). 4. If you read a frequency in both tests, check the connections for wires 444 and 445 at the Machine Control Board. Reference the machine wiring diagram to make sure they are in the correct terminal blocks and firmly seated.	The THCT communicates measured voltage from the plasma source back to the Machine Control Board using a differential frequency signal. This is much more immune to noise than a analog voltage.

11.5 TORCH HEIGHT CONTROL

11.5.1 Torch Cuts Too High

Cause: M210 Torch Height Control Voltage Set Too High		
Probability	How-To Steps	Need More?
High	➤ If using M200 AutoFS, ensure that you have selected the correct material and consumables on the Main tab of PathPilot. ➤ If you are manually specifying voltage using the M210 M-Code, verify that the P value in your G-Code is appropriate for the feed rate, cut current and consumables you are using according to the manufacturer's cut charts.	THC target voltage is set automatically when you select a material on the main PathPilot screen and use the M200 AutoFS function in your G-Code.
Cause: Incorrect Consumables Being Used		
Probability	How-To Steps	Need More?
Medium	➤ If you have both FineCut and standard consumables, verify that the correct nozzle is installed for the cut settings you have selected.	Standard consumables can require a THC voltage setting of up to 50V higher than FineCut consumables for the same cut current. Mixing up the two can produce a cut height error of more than 1" (25mm).
Cause: Bad Workpiece Clamp Connection		
Probability	How-To Steps	Need More?
High	➤ Make sure that the workpiece clamp is securely fastened to your material and plugged in to the plasma source. ➤ If you have your workpiece clamp cable routed through any terminal blocks, make sure that all connections are firmly tightened down.	A bad workpiece clamp connection can produce an additional voltage drop that will cause higher than usual THC voltage values. This will effect the cutting height.
Cause: Plasma Source Amperage Set Too High		
Probability	How-To Steps	Need More?
High	➤ If you are using the AutoFS Material Picker, make sure that you have your plasma source cut current set to match the value suggested on the main page of PathPilot. ➤ If you are using the Hypertherm control kit to allow PathPilot to control cut current and air pressure, verify that the adapter is connected to the controller's USB port and the RS485 cable is plugged in to the back of the plasma source.	

11: TROUBLESHOOTING

11.5 Torch Height Control

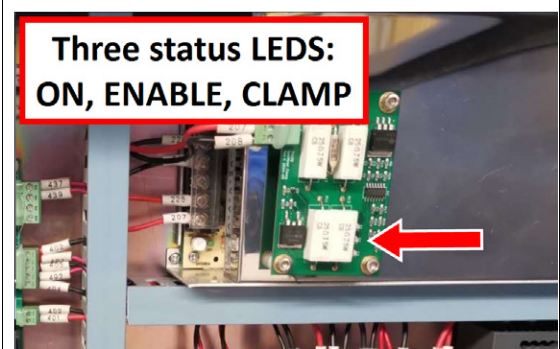
11.5.2 Torch Drags on the Material When Cutting

Cause: M208 Cut Height Set Too Low		
Probability	How-To Steps	Need More?
High	➤ If using M200 AutoFS, ensure that you have selected the correct material and consumables on the Main tab of PathPilot. ➤ If you are manually specifying a cut height in your G-Code with M208 make sure that the P value for height matches the consumables being used, cut current, and material.	After executing a G16 pierce, PathPilot will move the torch to the height specified by M208 or the AutoFS system. If this height is too low, Torch Height Control may not be able to activate and take over control of the Z axis because of safety limits on torch voltage.
Cause: Plasma Source Amperage Set Too Low		
Probability	How-To Steps	Need More?
High	➤ If you are using the AutoFS Material Picker, make sure that you have your plasma source cut current set to match the value suggested on the main page of PathPilot. ➤ If you are using the Hypertherm control kit to allow PathPilot to control cut current and air pressure, verify that the adapter is connected to the controller's USB port and the RS-485 cable is plugged in to the back of the plasma source.	

11: TROUBLESHOOTING

11.6 Servo Drives

Cause: Faulty Backfeed Clamp

Probability	How-To Steps	Need More?
Low	Check that the backfeed clamp is enabling itself. • Power on the main breaker, release the E-Stop button and press the blue Reset button. • Watch the status LEDs on the backfeed clamp. You should immediately see the ON LED.  Three status LEDs: ON, ENABLE, CLAMP • After about 1.5 sec, you should see the ENABLE LED light. If you do not see the ENABLE LED light, your backfeed clamp is not sensing input voltage and needs to be replaced.	

Cause: Backfeed Clamp Not Triggering

Probability	How-To Steps	Need More?
Low	Check that the backfeed clamp is triggering on hard braking. • Create a new G-Code program with the following test snippet: <pre>G1 Y20 X20 F1000 G1 Y0 X0 F1000 M99 (Repeat until Stop button is pressed)</pre> • Jog your machine to the -X, -Y corner and ZERO the X + Y axes and then run the test program.  WARNING! The test program will run continuously until the Stop button is pressed. Make sure you are in a safe position away from the motion envelope. • Watch the orange CLAMP LED on the backfeed clamp when the axes are slowing down. You should see a momentary flicker (<5 ms). • If you see no activity from the CLAMP LED and either axis faults during the test, replace your backfeed clamp.	The braking energy is only dissipated in the backfeed clamp under very high braking situations, which is why a G-Code program is used to test it instead of simply jogging the machine back and forth.

DIAGRAMS AND PARTS LISTS

IN THIS SECTION, YOU'LL LEARN:

- About this machine's components.

NOTICE! Only use Tormach-approved parts when making replacements. If you don't replace parts with those listed in this section, you may void your warranty.

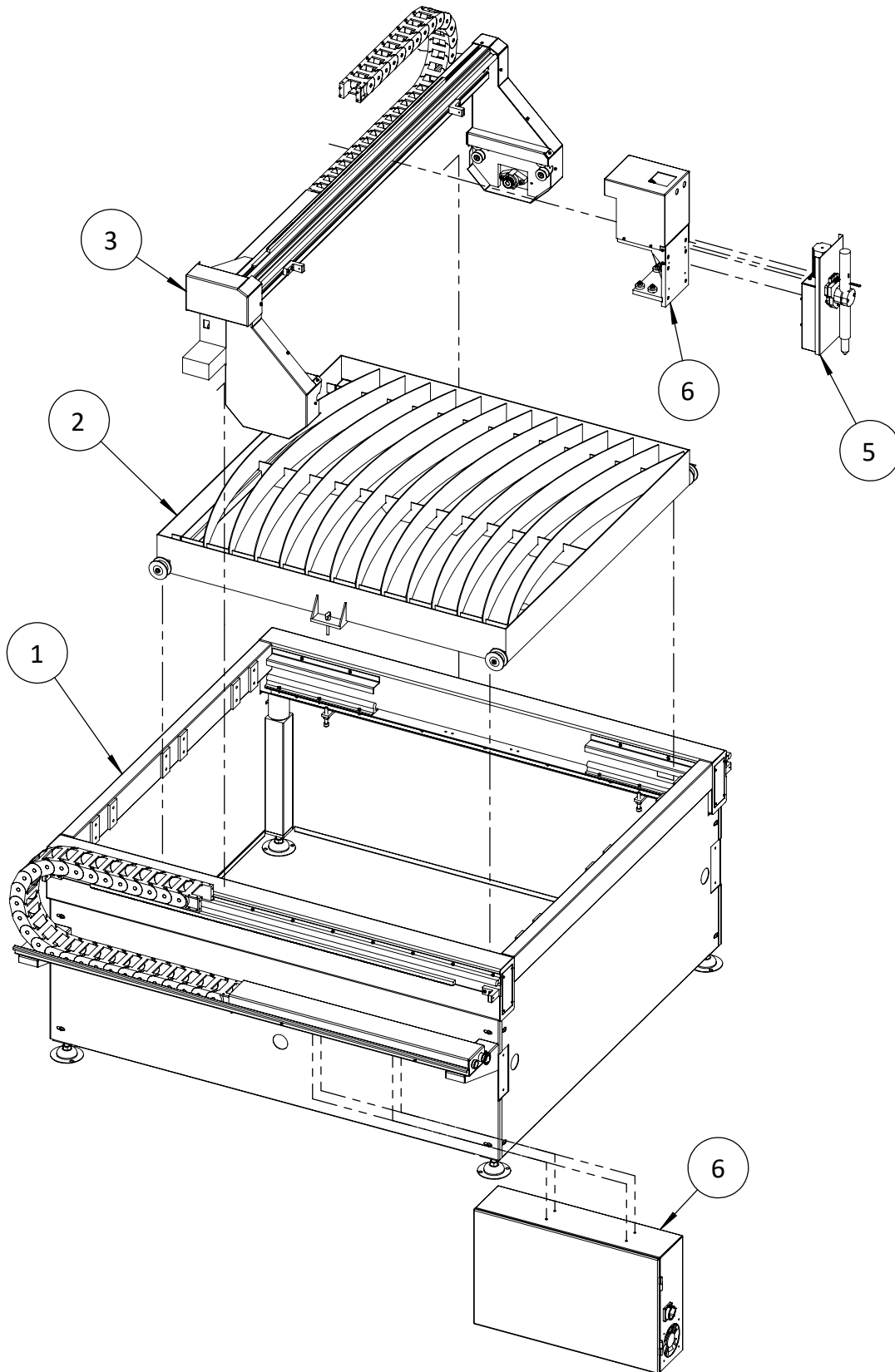
CONTENTS

12.1 Base Machine	180
12.2 Frame Assembly	182
12.3 Water Table	185
12.4 Y-Axis Gantry Assembly	187
12.5 X-Axis Carriage Assembly	190
12.6 Z-Axis Lifter Assembly	192
12.7 Electrical Cabinet Layout	195

12: DIAGRAMS AND PARTS LISTS

12.1 Base Machine

12.1 BASE MACHINE

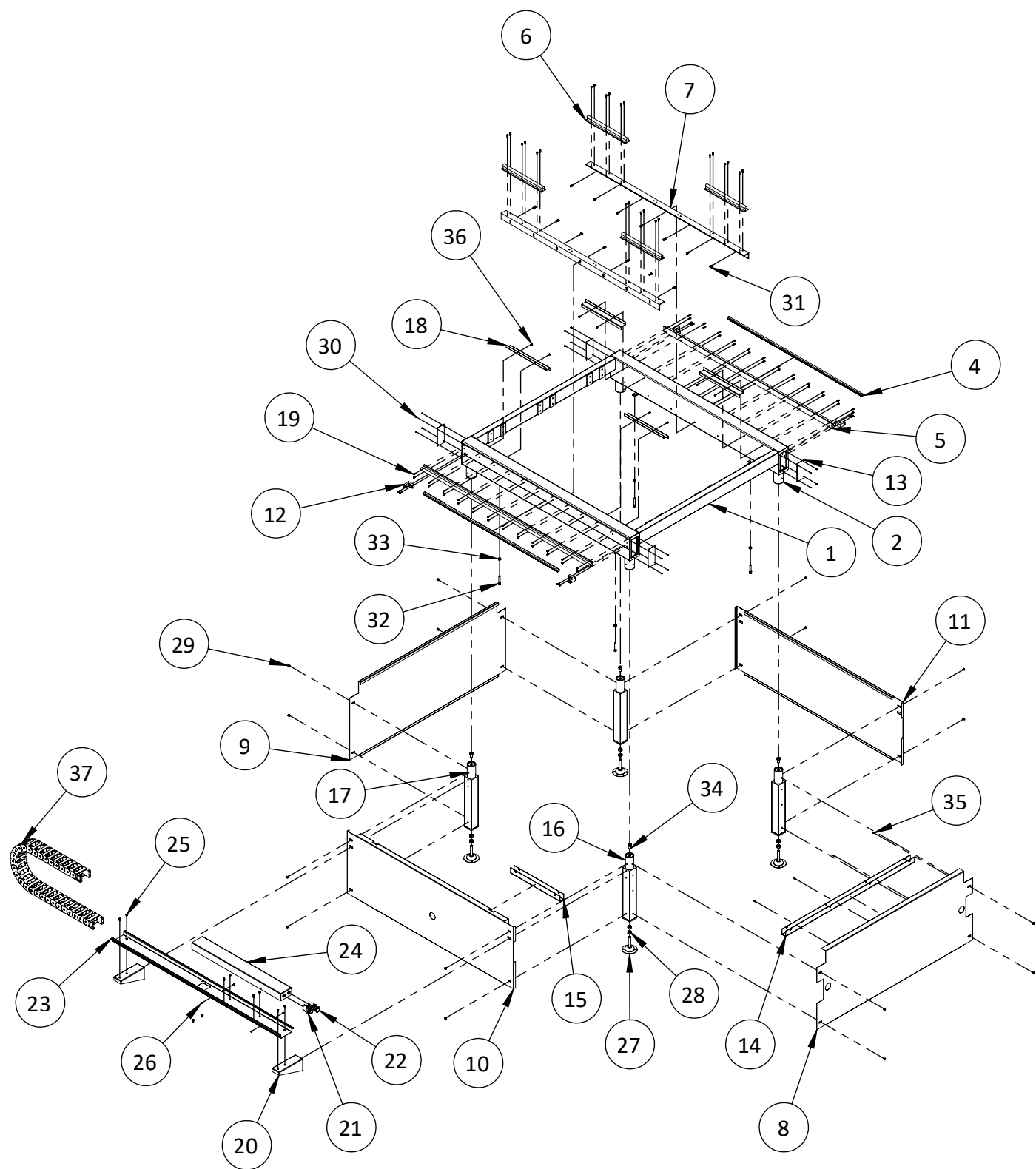


ID	Description	Quantity
1	<u>Frame Assembly (PN 50001)</u>	1
2	Electrical Cabinet Assembly (PN 50002)	1
3	<u>Water Table Assembly (PN 50003)</u>	1
4	<u>Y-Axis Gantry Assembly (PN 50004)</u>	1
5	<u>X-Axis Carriage Assembly (PN 50005)</u>	1
6	<u>Z-Axis Lifter Assembly (PN 50006)</u>	1

12: DIAGRAMS AND PARTS LISTS

12.2 Frame Assembly

12.2 FRAME ASSEMBLY



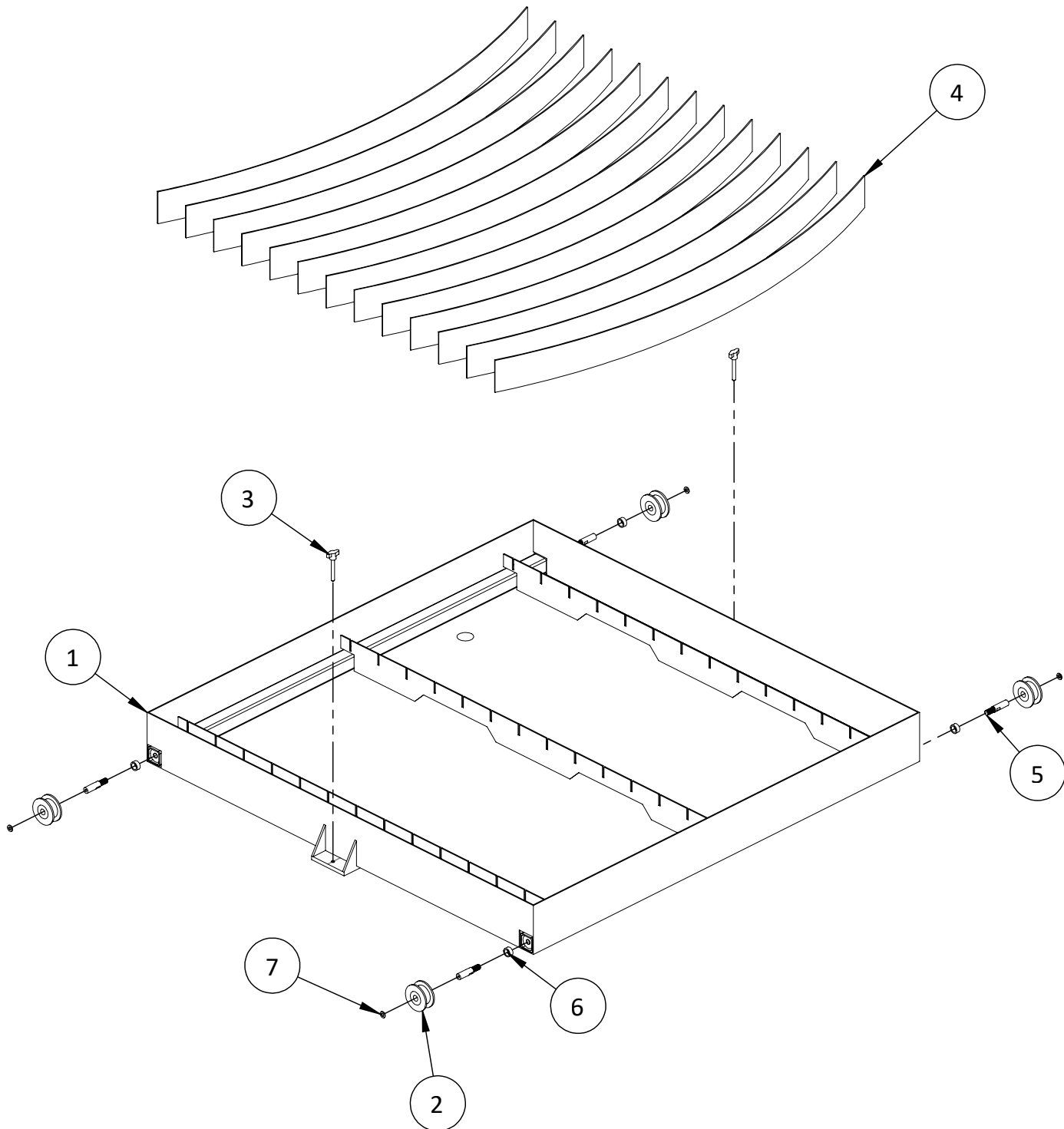
ID	Description	Quantity
1	Frame X Tube (PN 50010)	2
2	Frame Y Tube (PN 50011)	2
3	Grounding Bar (PN 39097)	1
4	Rack, Module 1, 1400 mm (PN 50047)	2
5	Linear Rail for 1300P Y (PN 50020)	2
6	Linear Rail, 20 mm (PN 50033)	4
7	Water Table Rail Support (PN 50129)	2
8	Front Panel Sheet Metal (PN 50015)	1
9	Rear Panel Sheet Metal (PN 50016)	1
10	Left Panel Sheet Metal (PN 50018)	1
11	Right Panel Sheet Metal (PN 50019)	1
12	Gantry Hard Stop, X-Axis (PN 50061)	4
13	Frame End Cap X (PN 50013)	4
14	Front Cable Tray (PN 50130)	1
15	Side Cable Tray (PN 50131)	1
16	1300P Frame Leg (PN 50009)	2
17	Frame Leg, B (PN 50133)	2
18	Water Table Track Shield (PN 50111)	4
19	Screw, Socket Head Cap, M05 × 0.8 × 015 (PN 20001)	72
20	Side Cable Tray Bracket (PN 50071)	2
21	Push Button (Blue LED) (PN 37342)	1
22	Emergency Stop Switch (PN 30462)	1
23	Side Cable Tray Base (PN 50021)	1
24	Side Cable Tray Top (PN 50022)	1
25	Screw, Socket Head Cap, M06 × 1 × 015 (PN 20001)	18
26	Screw, Socket Head Cap, M04 × 0.7 × 006 (PN 20001)	4
27	Machine Foot, Leveling, M16 × 2 Thread (PN 50119)	4
28	Nut, Hex, M16 (PN 20001)	8
29	Screw, Button Head Cap, M08 × 1.25 × 010 (PN 20001)	18
30	Screw, Socket Head Cap, M03 × 0.5 × 006 (PN 20001)	16
31	Screw, Socket Head Cap, M08 × 1.25 × 016 (PN 31681)	14
32	Screw, Socket Head Cap, M10 × 1.5 × 060 (PN 20001)	4

12: DIAGRAMS AND PARTS LISTS

12.2 Frame Assembly

ID	Description	Quantity
33	Nut, Hex, M10 (PN 36008)	4
34	Screw, Socket Head Cap, M16 × 1.75 × 025 (PN 20001)	4
35	Screw, Socket Head Cap, M04 × 0.7 × 010 (PN 30924)	4
36	Screw, Button Head Cap, M05 × 0.8 × 010 (PN 35774)	8
37	Energy Chain Link, Wide (PN 50017)	28
38	Energy Chain End Link, Wide, Internal Tabs (PN 50819)	1
39	Energy Chain End Link, Wide, External Tabs (PN 50820)	1

12.3 WATER TABLE

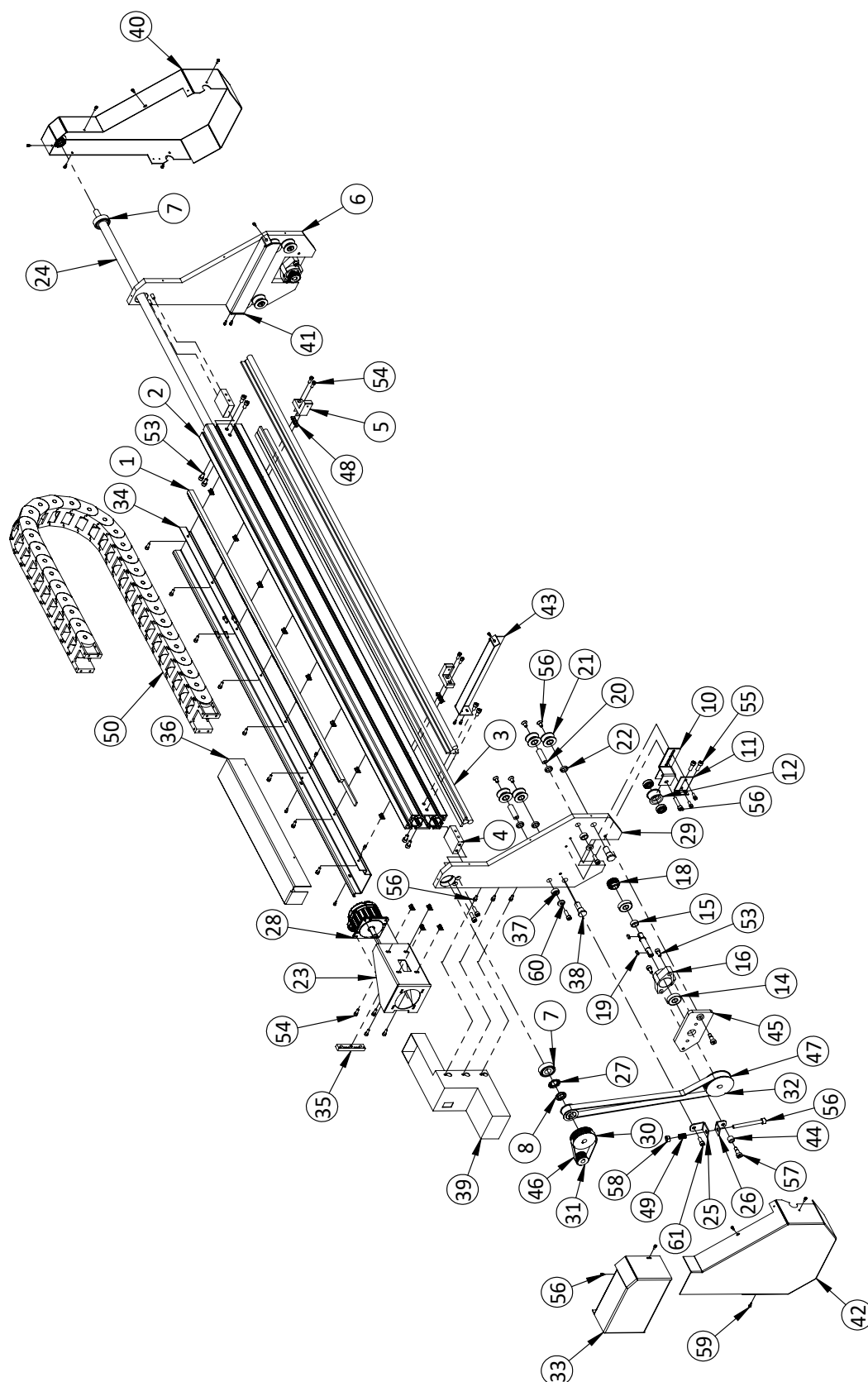


12: DIAGRAMS AND PARTS LISTS

12.3 Water Table

ID	Description	Quantity
1	Water Table Sheet Metal (PN 50030)	1
2	Water Table Roller (PN 50036)	4
3	Screw, Thumb, M8 × 1.25 - 60, 2 Arm Plastic Head (PN 50117)	2
4	Water Table Vertical Divider (PN 50032)	13
5	Water Table Roller Axle (PN 50118)	4
6	Water Table Roller Spacer (PN 50034)	4
7	Washer, Flat, M8 (PN 30531)	4

12.4 Y-AXIS GANTRY ASSEMBLY



12: DIAGRAMS AND PARTS LISTS

12.4 Y-Axis Gantry Assembly

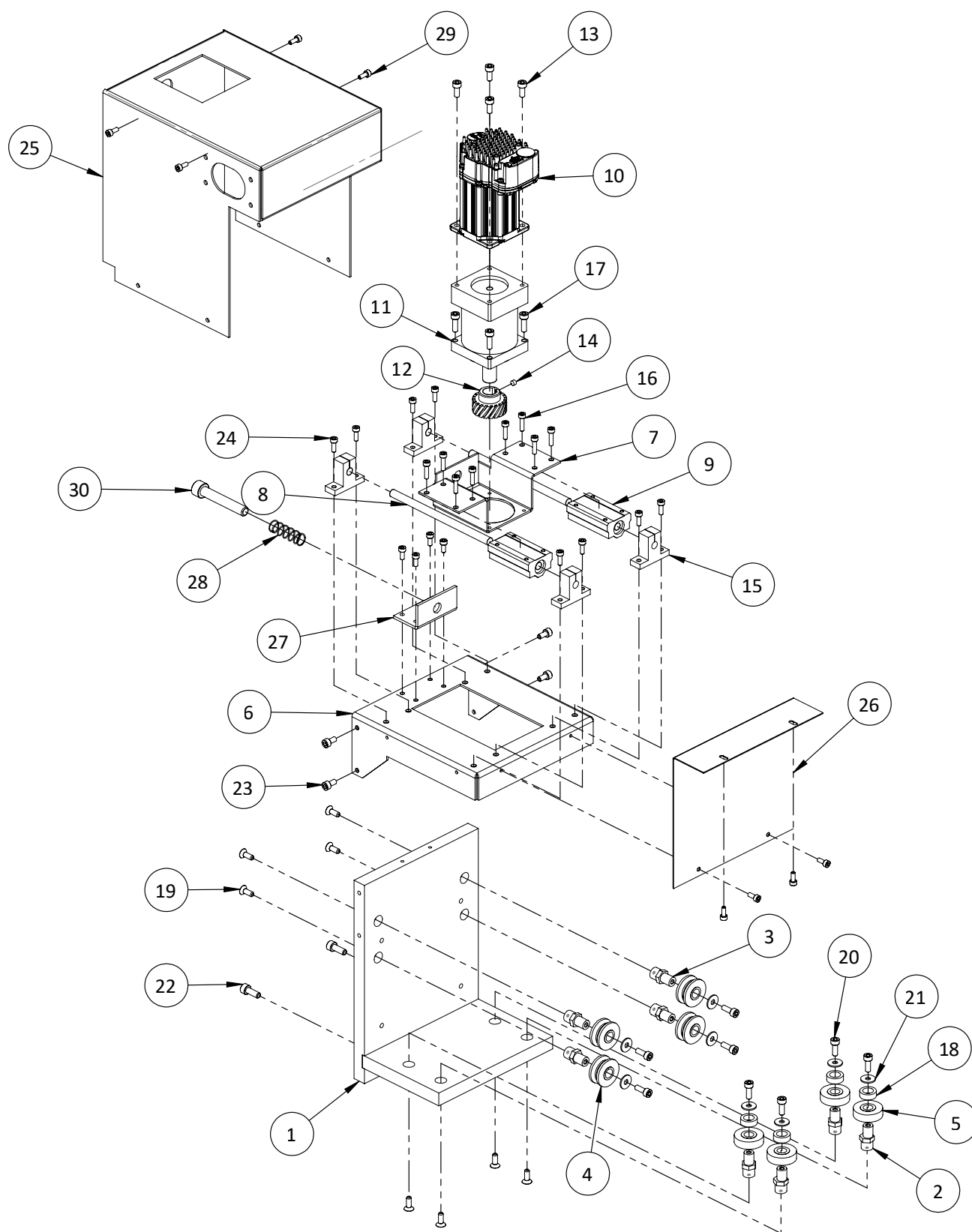
ID	Description	Quantity
1	Rack, Module 1, Plasma X-Axis (PN 50046)	1
2	Gantry Cross Beam Extrusion (PN 50044)	1
3	Linear Rail, 16 mm (PN 50045)	2
4	Cross Beam Mount Block (PN 50043)	2
5	Gantry Hard Stop, X-Axis (PN 50061)	2
6	Y-Axis Pulley Mount Plate X+ (PN 50037)	1
7	Y-Axis Drive Bearing (PN 50064)	2
8	Y-Axis Drive Lock Nut (PN 50063)	2
9	Timing Pulley, HTD-5, 25 Tooth, 14 mm Taper Lock (PN 50078)	2
10	Y-Axis Belt Tensioner (PN 50073)	2
11	Y-Axis Belt Tensioner L Bracket (PN 50134)	2
12	Y-Axis Belt Tensioner Wheel (PN 50074)	2
13	Y-Axis Belt Tensioner Wheel Bearing (PN 50135)	4
14	Y-Axis Pinion Bearing (PN 50137)	4
15	Y-Axis Pinion Bearing Spacer (PN 50138)	2
16	Y-Axis Pinion Mount Block (PN 50080)	2
17	Y-Axis Pinion Shaft (PN 50081)	2
18	Pinion, Module 1, 23T (PN 50072)	2
19	Key Stock, 5 mm × 15 mm (PN 50143)	4
20	Pin, 15 mm × 50 mm (PN 50049)	4
21	Linear Rail Roller, 20 mm Rail (PN 50050)	8
22	Y-Axis Roller Inner Spacer (PN 50069)	8
23	Y-Axis Motor Mount, NEMA 34 (PN 50082)	1
24	Y-Axis Drive Shaft (PN 50062)	1
25	Y-Axis Rack and Pinion Tensioner Bracket, Top (PN 50052)	2
26	Y-Axis Rack and Pinion Tensioner Bracket, Bottom (PN 50051)	2
27	Y-Axis Drive Lock Washer (PN 50065)	2
28	Plasma Y-Axis Servo, Clearpath CPM-SDSK-3411S-ELN (PN 39104)	1
29	Y-Axis Pulley Mount Plate X- Mirrored (PN 50038)	1
30	Timing Pulley, HTD-5, 40 Tooth, 14 mm Taper Lock (PN 50076)	1
31	Timing Pulley, HTD-5, 25 Tooth, 14 mm ID (PN 50077)	1
32	Timing Pulley, HTD-5, 50 Tooth, 14 mm Shaft (PN 50079)	2

ID	Description	Quantity
33	Y-Axis Motor Pulley Cover (PN 50056)	1
34	Gantry Cable Tray Base (PN 50059)	1
35	Grounding Bar (PN 39097)	1
36	Gantry Cable Tray Cover (PN 50060)	1
37	Y-Axis Roller Outer Spacer (PN 50070)	4
38	Y-Axis Roller Eccentric Adjust (PN 50066)	4
39	Gantry Cable Tray Connector (PN 50041)	1
40	Y-Axis Pulley Cover, X+ (PN 50039)	1
41	Y-Axis Linear Roller Cover, X+ (PN 50140)	1
42	Y-Axis Pulley Cover, X- (PN 50040)	1
43	Y-Axis Linear Roller Cover, X- (PN 50141)	1
44	Y-Axis Tensioner Bracket Retainer (PN 50142)	2
45	Y-Axis Tensioner Pinion Mount (PN 50057)	2
46	Timing Belt, HTD-5, Motor to Drive Shaft (PN 50112)	1
47	Timing Belt, HTD-5, Drive Shaft to Pinion (PN 50113)	1
48	T-Nut, M6, 8 mm WD (PN 50048)	16
49	Compression Spring, 10.5 mm ID, 20 mm LG (PN 50053)	2
50	Energy Chain Link, Narrow (PN 50055)	29
51	Energy Chain End Link, Narrow, External (PN 50817)	1
52	Energy Chain End Link, Narrow, Internal Tabs (PN 50818)	1
53	Screw, Socket Head Cap, M8 × 1.25 - 16 (PN 31681)	12
54	Screw, Socket Head Cap, M6 × 1 - 16 (PN 33117)	26
55	Screw, Socket Head Cap, M8 × 1.25 - 25 (PN 31618)	12
56	Screw, Socket Head Cap, M4 × 0.7 - 6 (PN 20001)	17
57	Screw, Shoulder, M6 × 1 - 10 (PN 35926)	4
58	Nut, Hex, M10 (PN 36008)	2
59	Screw, Socket Head Cap, M4 × 0.7 - 10 (PN 30924)	19
60	Washer, Flat, M8 (PN 50821)	4
61	Screw, Socket Head Cap, M8 × 1.25 - 20 (PN 31895)	2

12: DIAGRAMS AND PARTS LISTS

12.5 X-Axis Carriage Assembly

12.5 X-AXIS CARRIAGE ASSEMBLY



12: DIAGRAMS AND PARTS LISTS

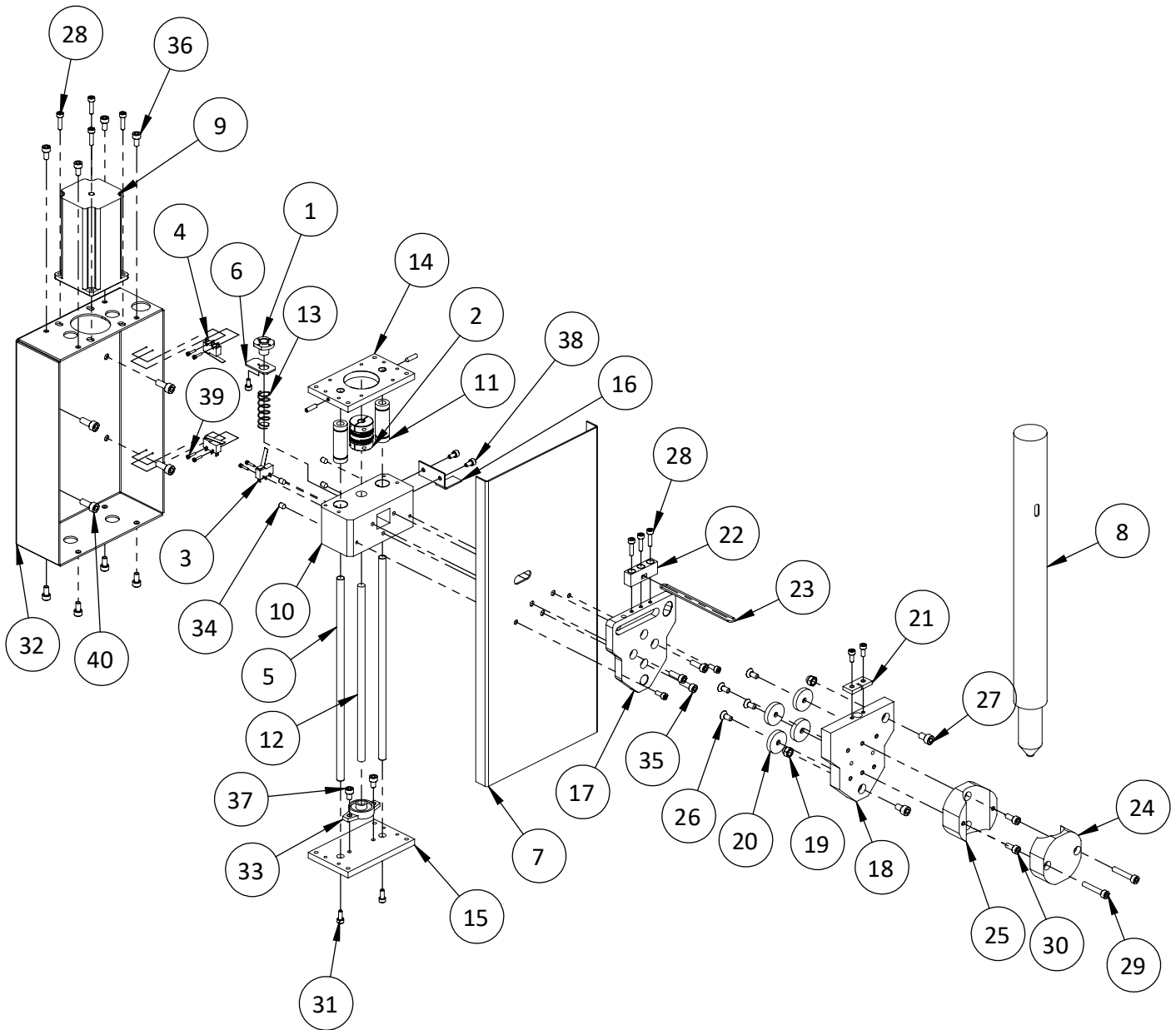
12.5 X-Axis Carriage Assembly

ID	Description	Quantity
1	X-Axis Carriage (PN 50083)	1
2	X Linear Rail Roller Shaft Support Plate, Eccentric (PN 50087)	4
3	X Linear Rail Roller Shaft, Support Plate (PN 50086)	4
4	Linear Rail Roller, 16 mm Rail (PN 50092)	4
5	Linear Rail Roller, 16 mm Rail Flat (PN 50093)	4
6	X-Axis Linear Rail Mounting Bracket (PN 50149)	1
7	X-Axis Motor Bracket (PN 50115)	1
8	Lead Screw, 8 mm × 170 mm (PN 50094)	2
9	Linear Bearing, 8 mm (PN 50102)	2
10	X-Axis Motor, ClearPath CPM-SDSK-2321S-EQN (PN 39105)	1
11	Planetary Gear Box, NEMA 23, 5:1 (PN 50088)	1
12	Pinion, Module 1, 23T (PN 50072)	1
13	Screw, Socket Head Cap, M5 × 0.8 - 12 mm (PN 31353)	4
14	Screw, Cone Point Set, M5x5 120° (PN 20001)	1
15	Linear Rail Clamp, 8 mm (PN 50101)	4
16	Screw, Socket Head Cap, M4 × 0.7 - 16 (PN 37751)	8
17	Screw, Socket Head Cap, M5 × 0.8 - 16 (PN 30546)	4
18	X-Axis Linear Roller Washer (PN 50148)	4
19	Screw, Socket Head Countersunk, M5 × 0.8 - 14 (PN 20001)	8
20	Screw, Socket Head Cap, M5 × 0.8 - 14 (PN 31563)	8
21	Washer, Flat, M5 (PN 50822)	8
22	Screw, Socket Head Cap, M6 × 1 - 16 (PN 33117)	2
23	Screw, Socket Head Cap, M5 × 0.8 - 10 (PN 31641)	4
24	Screw, Socket Head Cap, M4 × 0.7 - 12 (PN 31360)	8
25	X Carriage Top Sheet Metal (PN 50085)	1
26	X Carriage Back Cover Plate (PN 50084)	1
27	X-Axis Tensioner Bracket (PN 50116)	1
28	Z-Axis Compression Spring (PN 50122)	1
29	Screw, Socket Head Cap, M4 × 0.7 - 10 (PN 30924)	12
30	Screw, Socket Head Cap, M10 × 1.5 - 60 (PN 20001)	1

12: DIAGRAMS AND PARTS LISTS

12.6 Z-Axis Lifter Assembly

12.6 Z-AXIS LIFTER ASSEMBLY



12: DIAGRAMS AND PARTS LISTS

12.6 Z-Axis Lifter Assembly

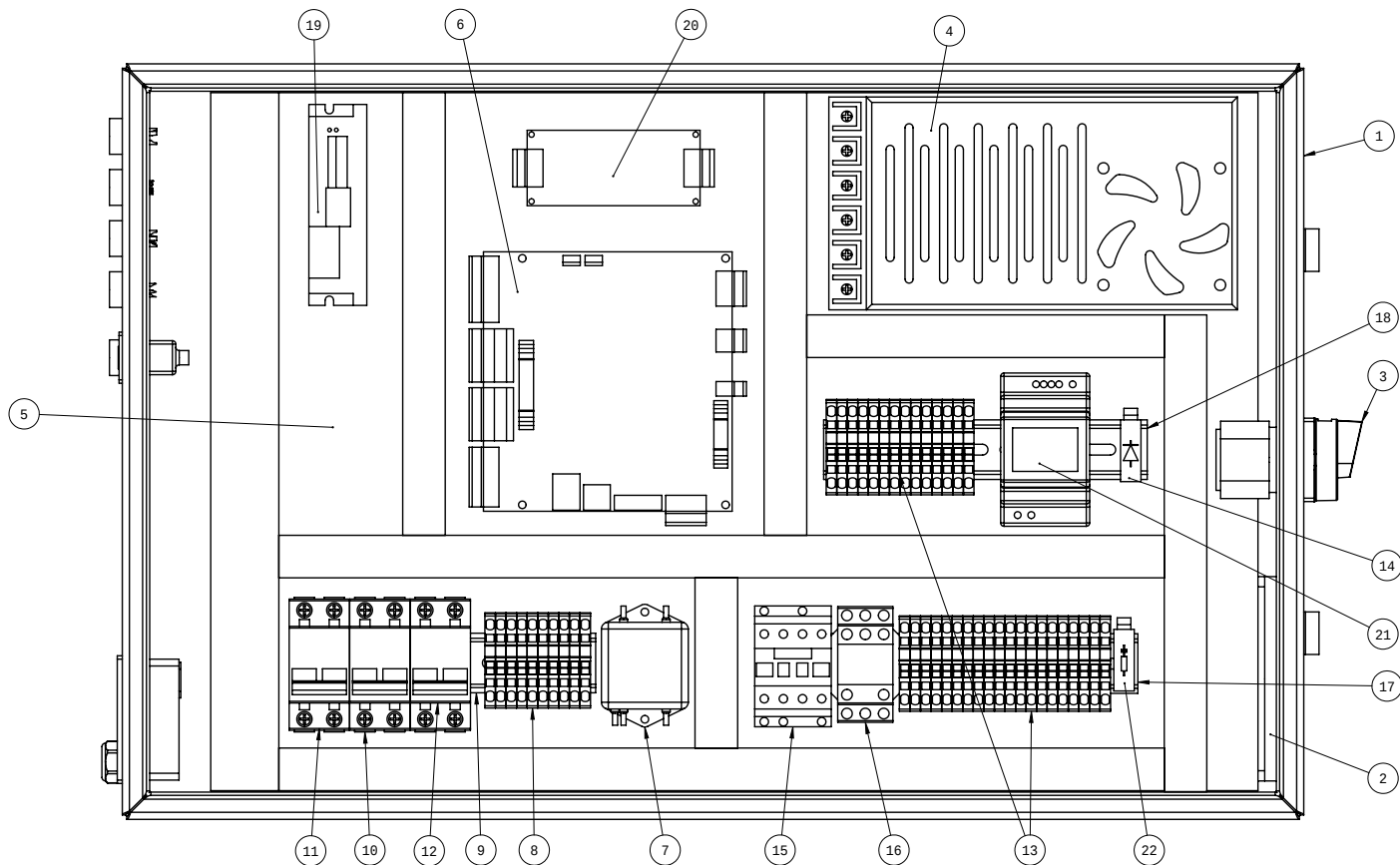
ID	Description	Quantity
1	Lead Screw Nut, 8 mm (PN 50107)	1
2	Lead Screw Coupler, 8 mm (PN 50108)	1
3	Z-Axis Limit Switch Vertical Trigger (PN 50105)	1
4	Z-Axis Limit Switch (PN 50104)	2
5	Linear Rail, 8 mm × 225 mm (PN 50100)	2
6	Z-Axis Lifter Touch Switch Trigger Plate (PN 50103)	1
7	Z-Axis Motor Mount Front Cover (PN 50099)	1
8	Duramax Torch	1
9	Stepper, NEMA 23, 3 Stack, 8 mm Shaft (PN 39106)	1
10	Z-Axis Linear Bearing Carrier (PN 50120)	1
11	Linear Bearing, Round, 10 mm (PN 50146)	2
12	Lead Screw, 8 mm × 170 mm (PN 50106)	1
13	Z-Axis Compression Spring (PN 50122)	1
14	Z-Axis Upper Linear Rail Mounting Plate (PN 50124)	1
15	Z-Axis Lower Linear Rail Mounting Plate (PN 50125)	1
16	Z-Axis Limit Switch Flag (PN 50121)	1
17	Magnetic Breakaway Base (PN 50096)	1
18	Magnetic Breakaway Kinematic Plate (PN 50126)	1
19	Magnetic Breakaway Torch Holder Acorn Nut (PN 50703)	2
20	Magnetic Breakaway Torch Holder Magnet (PN 50702)	4
21	Magnetic Breakaway Torch Holder Tether Clamp (PN 50701)	1
22	Magnetic Breakaway Torch Holder Tether Base (PN 50700)	1
23	Magnetic Breakaway Torch Holder Tether (PN 50704)	1
24	Magnetic Breakaway Torch Top Clamp (PN 50699)	1
25	Magnetic Breakaway Torch Bottom Clamp (PN 50150)	1
26	Screw, Socket Head Countersunk, M5 × 0.8 - 12 (PN 38894)	4
27	Screw, Socket Head Cap, M6 × 1 - 10 (PN 20001)	2
28	Screw, Socket Head Cap, M4 × 0.7 - 16 (PN 37751)	7
29	Screw, Socket Head Cap, M5 × 0.8 - 30 (PN 31602)	2
30	Screw, Socket Head Cap, M5 × 0.8 - 12 mm (PN 31353)	2
31	Screw, Socket Head Cap, M4 × 0.7 - 10 (PN 30924)	6
32	Z-Axis Motor Mount Sheet Metal (PN 50098)	1

12: DIAGRAMS AND PARTS LISTS

12.6 Z-Axis Lifter Assembly

ID	Description	Quantity
33	Lead Screw End Support (PN 50123)	1
34	Screw, Cone Point Set, M5 × 8 90° (PN 30337)	4
35	Screw, Socket Head Cap, M5 × 0.8 - 16 (PN 30546)	3
36	Screw, Socket Head Cap, M5 × 0.8 - 10 (PN 31641)	8
37	Screw, Socket Head Cap, M5 × 0.8 - 8 (PN 30844)	2
38	Screw, Socket Head Cap, M4 × 0.7 - 8 (PN 30891)	3
39	Screw, Socket Head Cap, M2 × 0.4 - 10 (PN 20001)	6
40	Screw, Socket Head Cap, M6 × 1 - 16 (PN 33117)	4
41	Screw, Cone Point Set, M4 × 16 90° (PN 20001)	2

12.7 ELECTRICAL CABINET LAYOUT



12: DIAGRAMS AND PARTS LISTS

12.7 Electrical Cabinet Layout

ID	Description	Quantity
1	Cabinet (PN 50024)	1
2	Fan (24 Vdc, 120 mm) (PN 37526)	1
3	Main Disconnect Switch (PN 30454)	1
4	Power Supply, 90-132 Vac / 180-264 Vac, 75 Vdc, 600 W, 8 A (PN 39101)	1
5	Sheet Metal Panel (PN 50027)	1
6	ECM1 v1.5, Machine Control Board (PN 37509)	1
7	Filter (110 Vac/250 Vac, 10 A) (PN 32350)	1
8	Ground Terminal Block (Screwless, DIN Mount) (PN 34130)	10
9	DIN Rail (35 × 7.5), 170 mm Long (PN 37910)	1
10	Circuit Breaker (10 A, 2 Pole) (PN 37346)	1
11	Circuit Breaker (20 A, 2 Pole) (PN 37345)	2
12	Circuit Breaker (20 A, 2 Pole) (PN 37345)	
13	Terminal Block (Screwless, DIN Mount) (PN 34127)	34
14	DC Flyback Protection Diode (PN 37514)	1
15	Contactor (24 Vdc Coil, 1 Auxiliary Pole) (PN 37343)	1
16	Relay (24 Vdc Coil, DIN Mount) (PN 37515)	1
17	DIN Rail (35 × 7.5), 225 mm Long (PN 37910)	1
18	DIN Rail (35 × 7.5), 190 mm Long (PN 37910)	1
19	Stepper Driver (PN 38686)	1
20	Torch Height Control Board (PN 39096)	1
21	Power Supply (24 Vdc, 60 W) (PN 50461)	1
22	Electrical Noise Suppressor (DIN Mount) (PN 37358)	1

ELECTRICAL SCHEMATICS

IN THIS SECTION, YOU'LL LEARN:

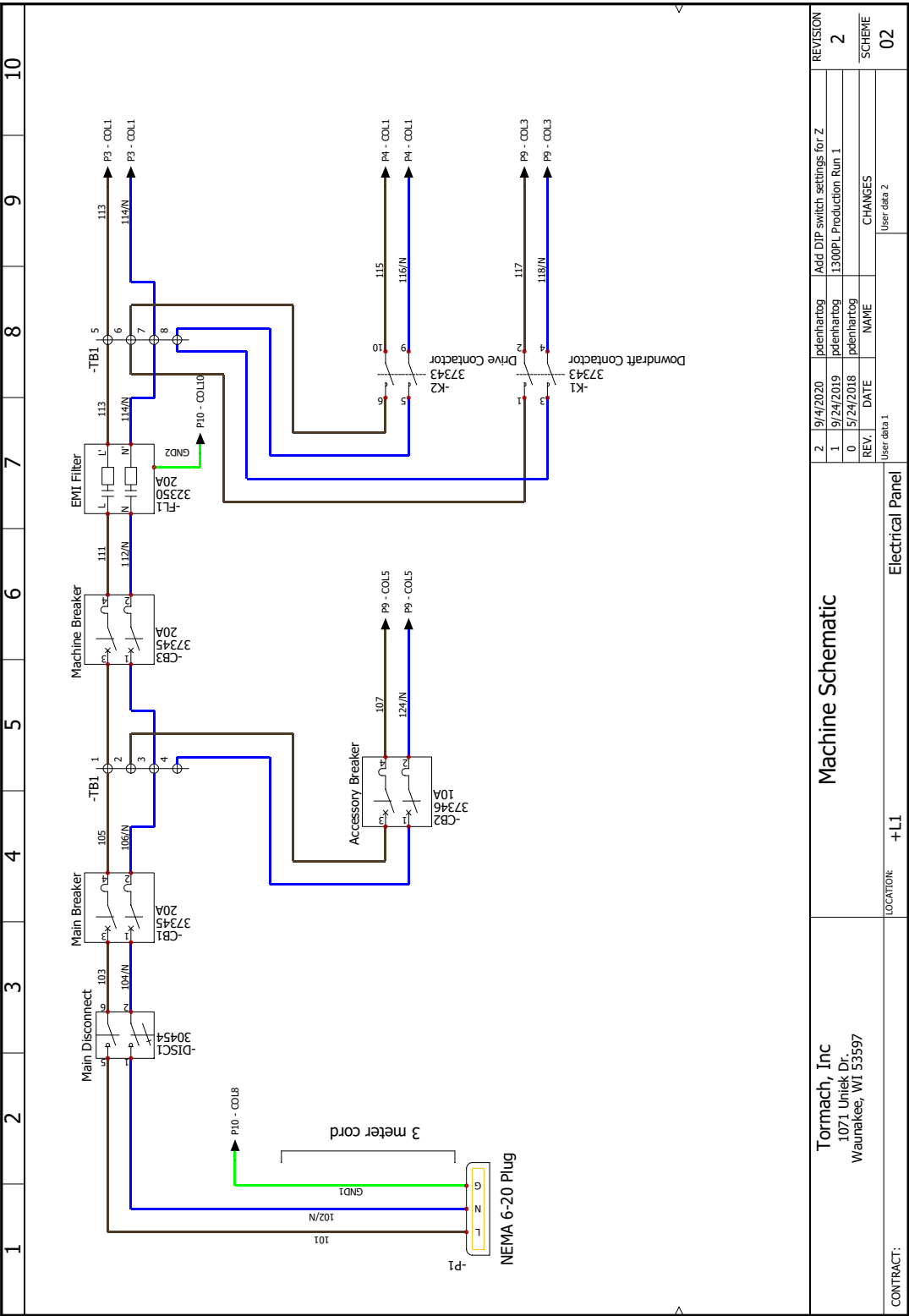
- About the electrical schematics for this machine's electronics.

CONTENTS

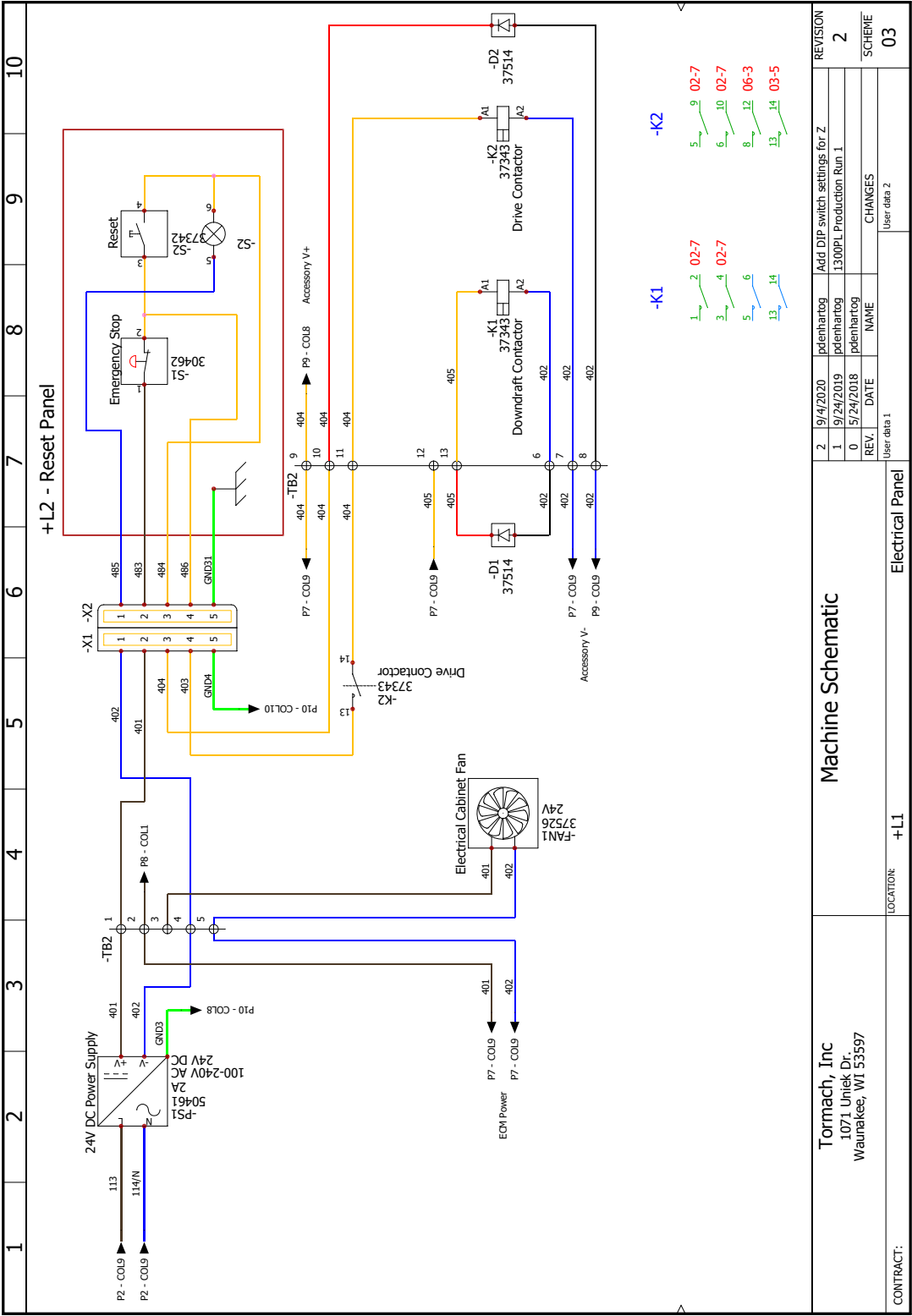
13.1 230 Vac Power (Sheet 2)	198
13.2 24 Vdc Controls (Sheet 3)	199
13.3 Axis Drive Control (Sheet 4)	200
13.4 A-Axis Driver (Sheet 5)	201
13.5 Plasma Source Control (Sheet 6)	202
13.6 Machine Control Board (Sheet 7)	203
13.7 Torch Limit and Breakaway Switches (Sheet 8)	204
13.8 Accessory and Auxiliary Power (Sheet 9)	205
13.9 Grounds (Sheet 10)	206
13.10 Terminal Strips (Sheets 11-16)	207

13: ELECTRICAL SCHEMATICS
13.1 230 Vac Power (Sheet 2)

13.1 230 VAC POWER (SHEET 2)



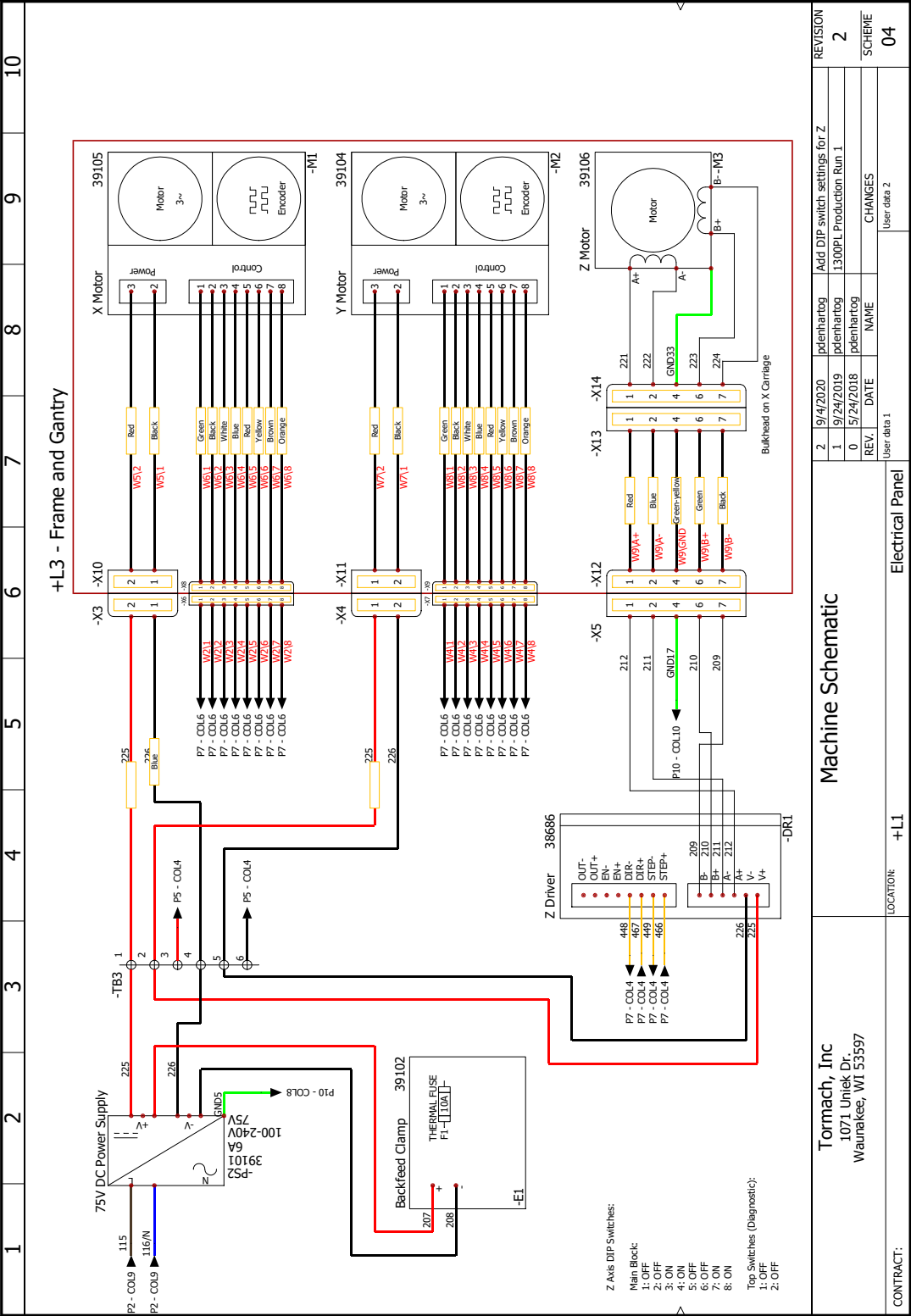
13.2 24 VDC CONTROLS (SHEET 3)



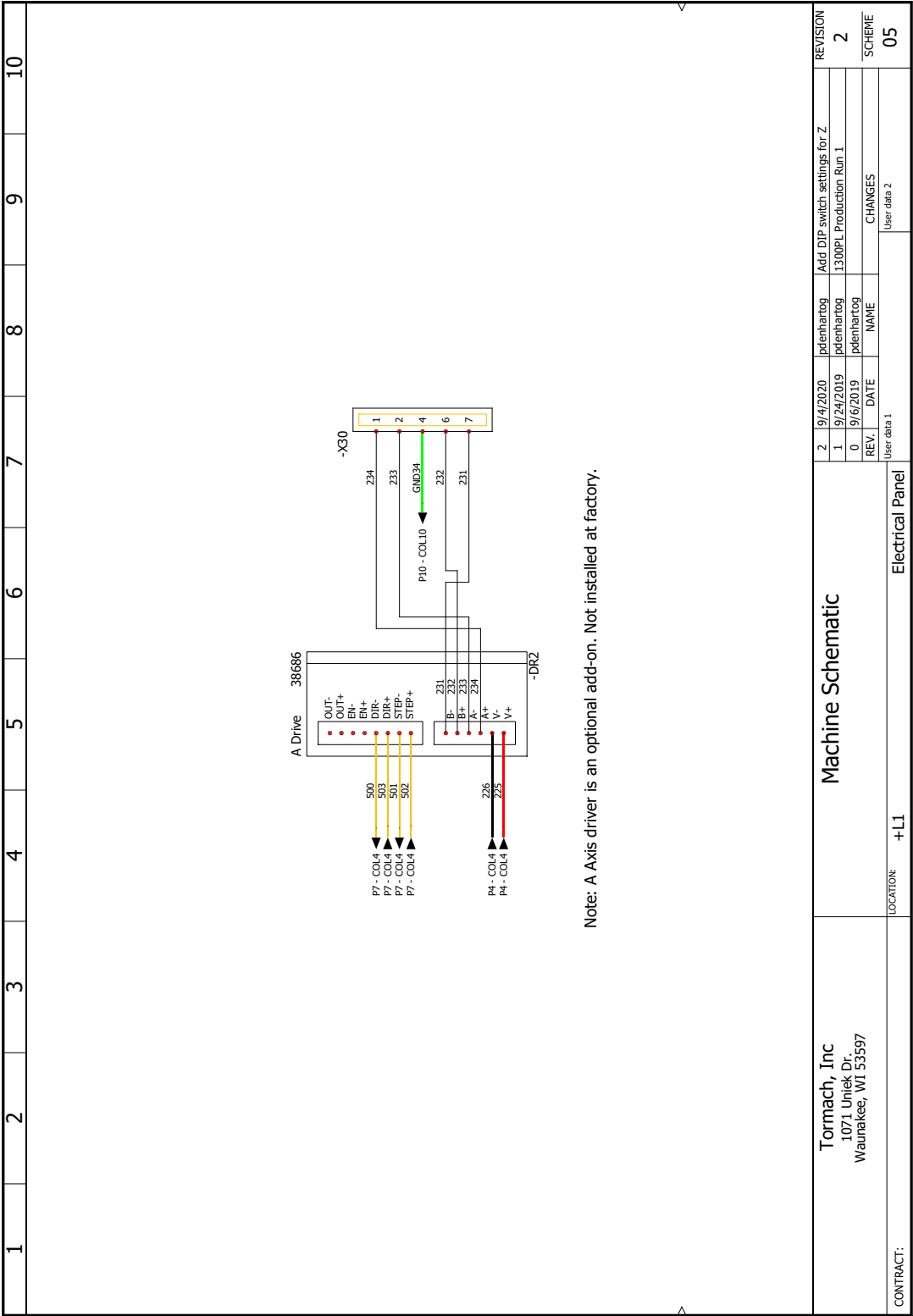
SOLIDWORKS Electrical

13: ELECTRICAL SCHEMATICS
13.3 Axis Drive Control (Sheet 4)

13.3 AXIS DRIVE CONTROL (SHEET 4)



13.4 A-AXIS DRIVER (SHEET 5)

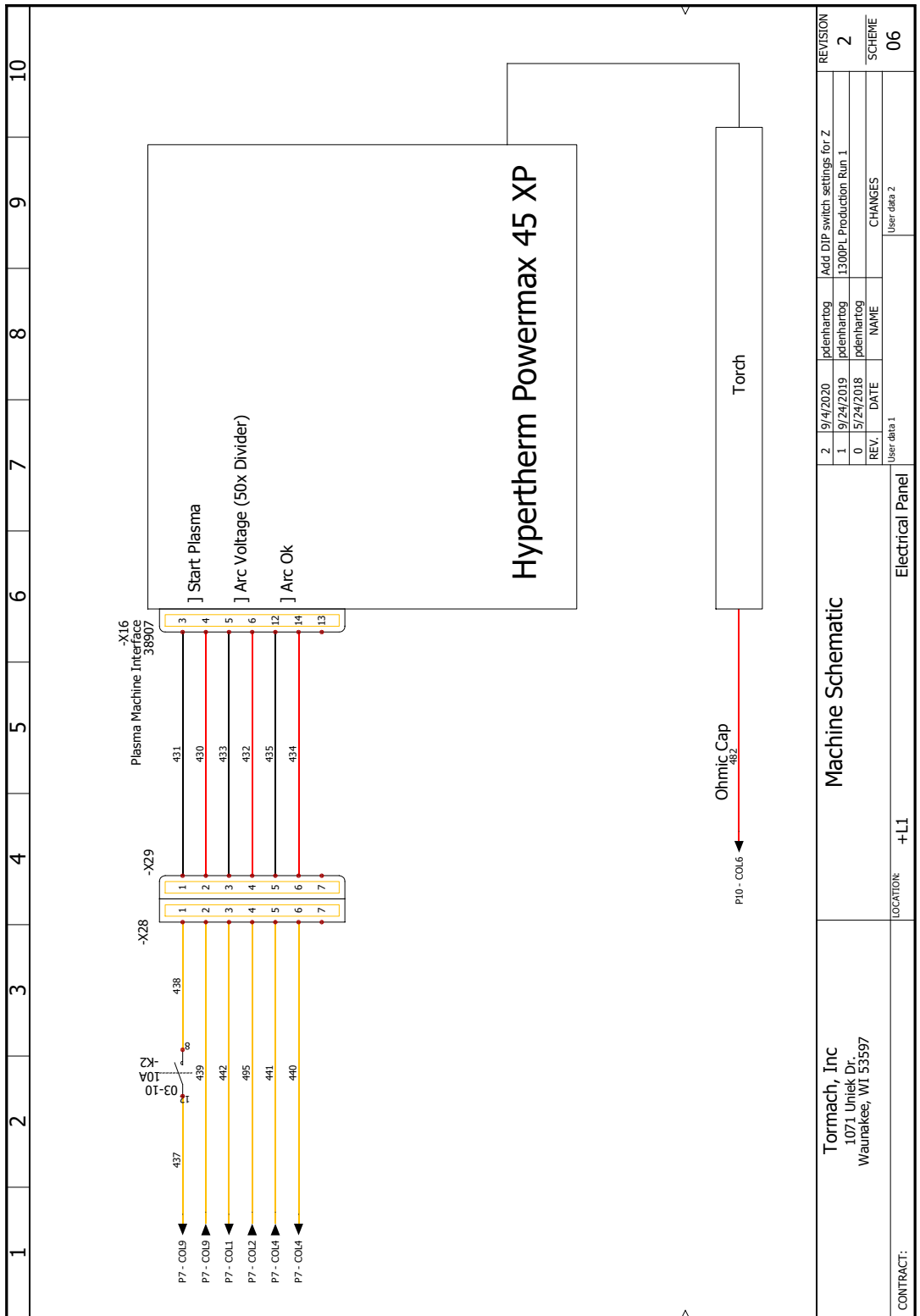


SOLIDWORKS Electrical

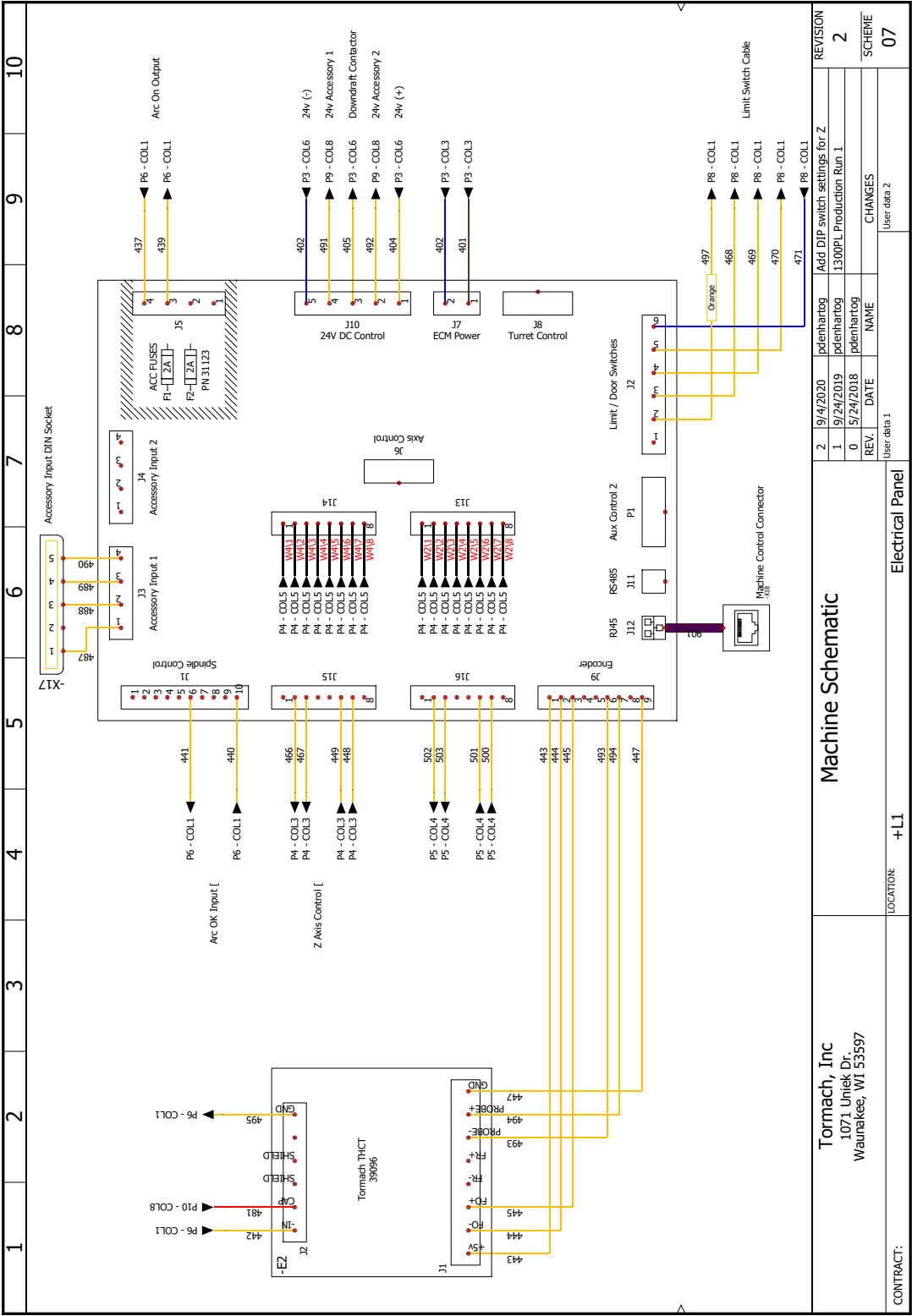
Document realized with version : 2018.0.2.1.4

13.5 Plasma Source Control (Sheet 6)

13.5 PLASMA SOURCE CONTROL (SHEET 6)



13.6 MACHINE CONTROL BOARD (SHEET 7)



SOLIDWORKS Electrical

Machine Schematic

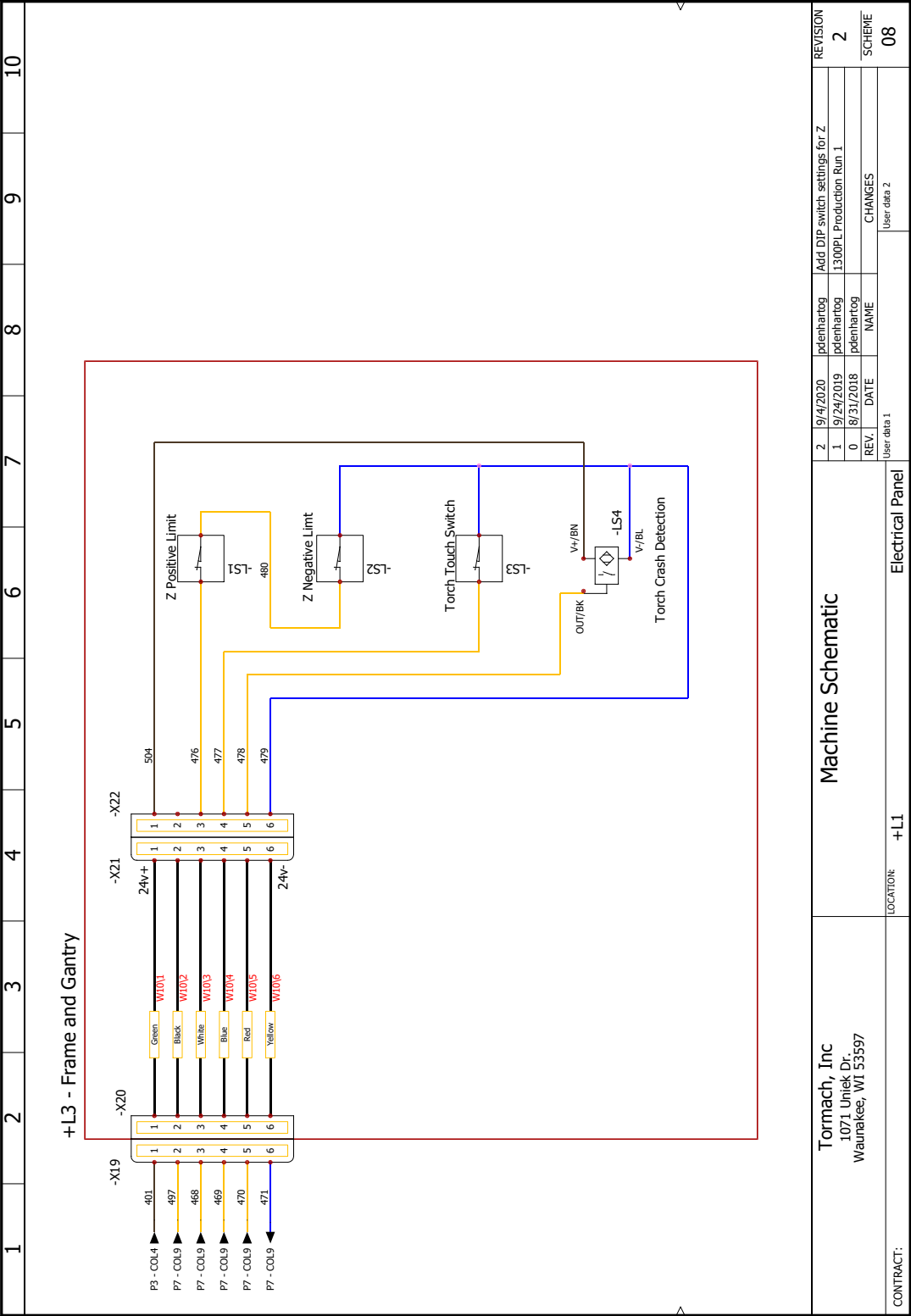
Tormach, Inc
1071 Uniek Dr
Waunakee, WI 53597

CONTRACT:		LOCATION: +L1		Electrical Panel	
REVISION		REV.		DATE	
2		1		9/4/2020	
2		0		5/24/2018	
1300PL Production Run 1		Add DIP switch settings for Z		User data 1	
CHANGES		NAME		User data 2	
SCHEME		07			

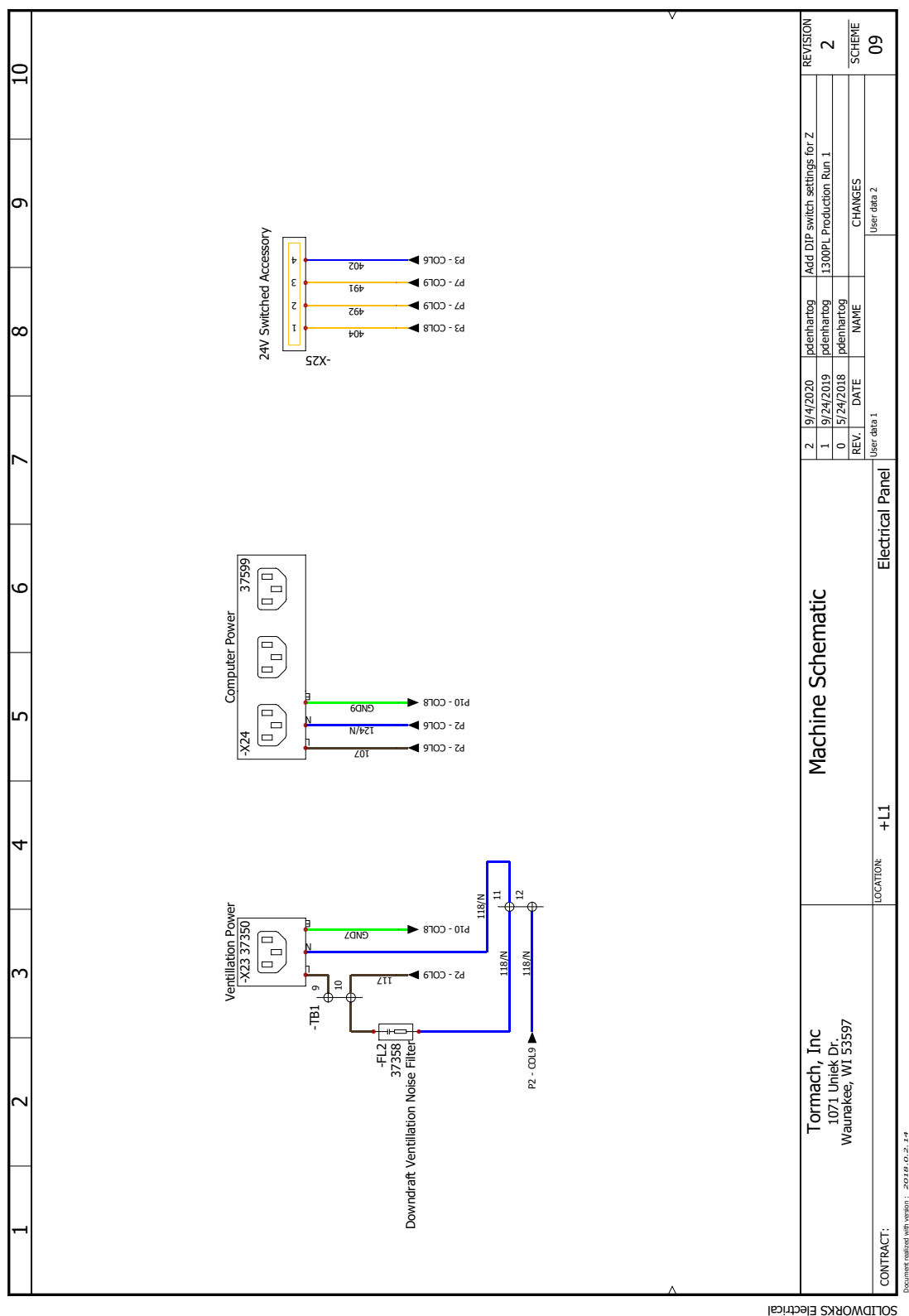
Document created with version: 2018.0.2.1.4

13: ELECTRICAL SCHEMATICS
13.7 Torch Limit and Breakaway Switches (Sheet 8)

13.7 TORCH LIMIT AND BREAKAWAY SWITCHES (SHEET 8)

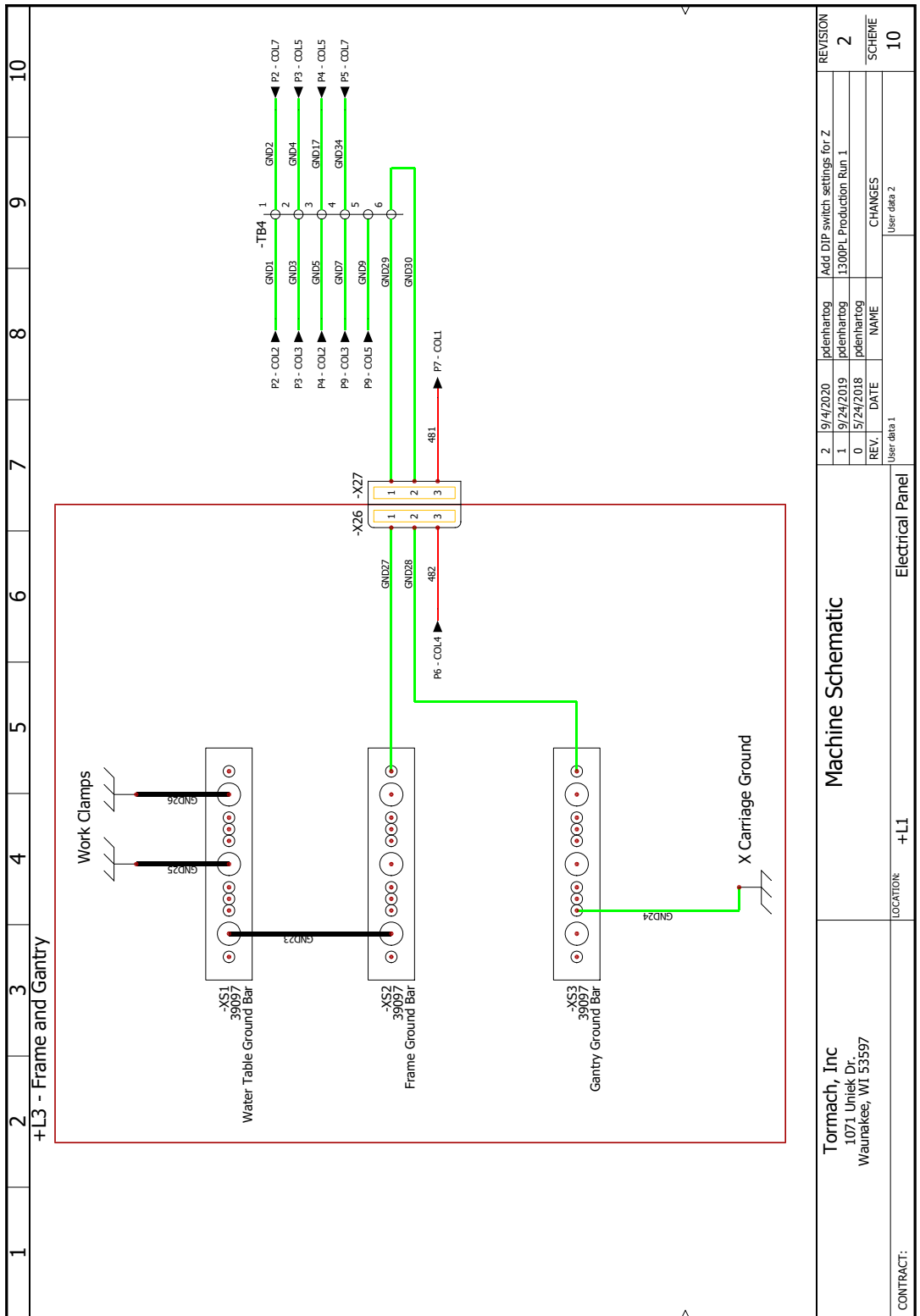


13.8 ACCESSORY AND AUXILIARY POWER (SHEET 9)



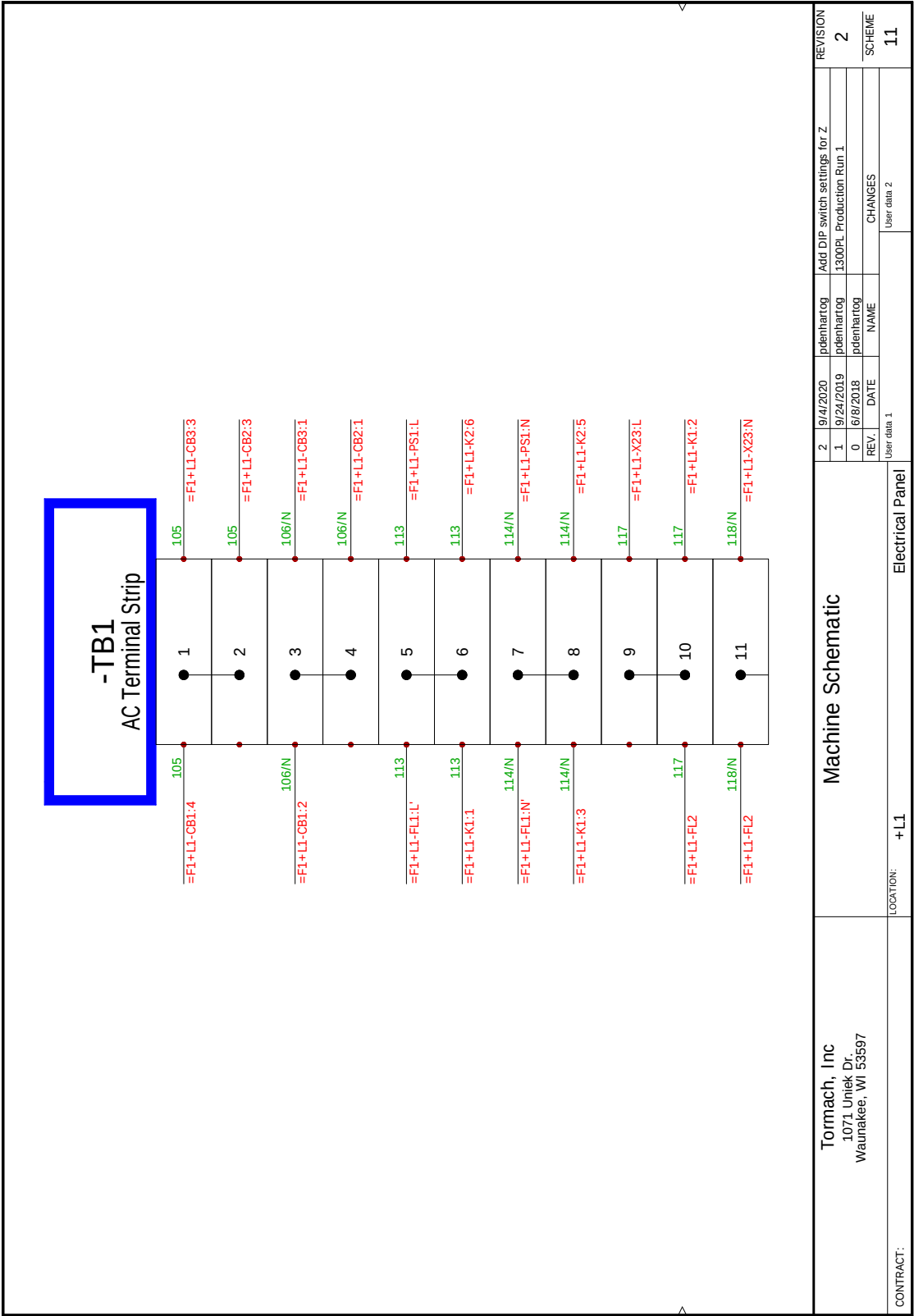
13.9 Grounds (Sheet 10)

13.9 GROUNDS (SHEET 10)



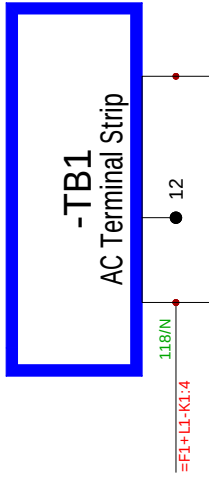
13.10 TERMINAL STRIPS (SHEETS 11-16)

13.10.1 TB1: AC Terminal Strip (Sheet 11)

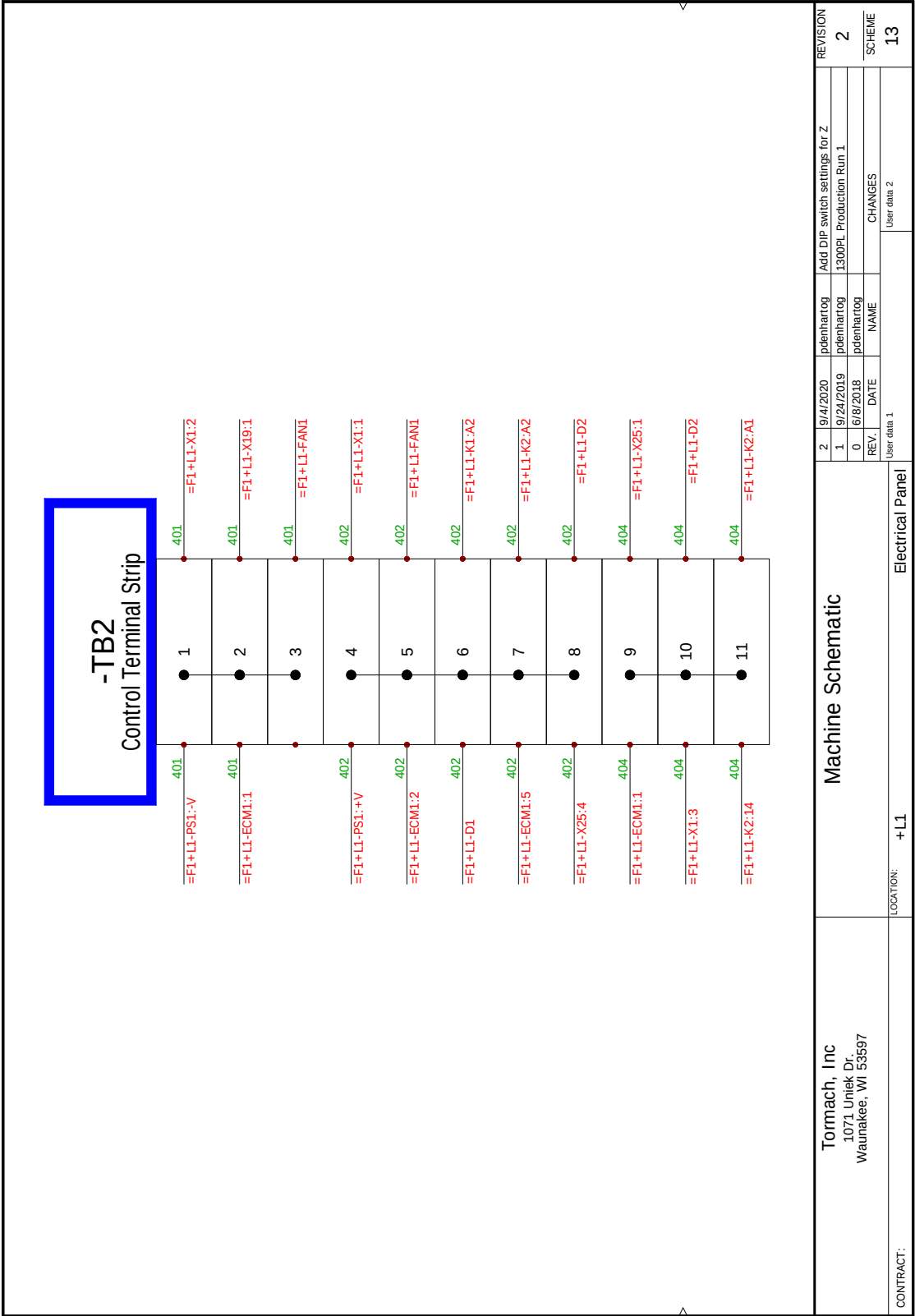


13.10 Terminal Strips (Sheets 11-16)

13.10.2 TB1: AC Terminal Strip (Sheet 12)

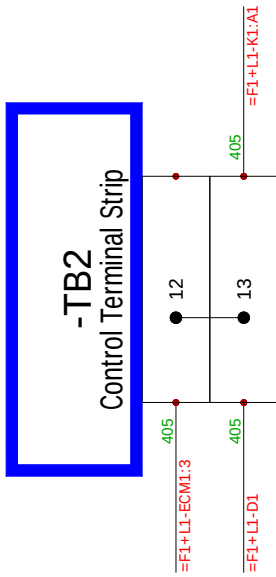
[illegible]

13.10.3 TB2: Control Terminal Strip (Sheet 13)

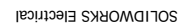


13.10 Terminal Strips (Sheets 11-16)

13.10.4 TB2: Control Terminal Strip (Sheet 14)



13.10.5 TB3: 75 Vdc Terminal Strip (Sheet 15)



13.10 Terminal Strips (Sheets 11-16)

13.10.6 TB4: Ground Terminal Strip (Sheet 16)

