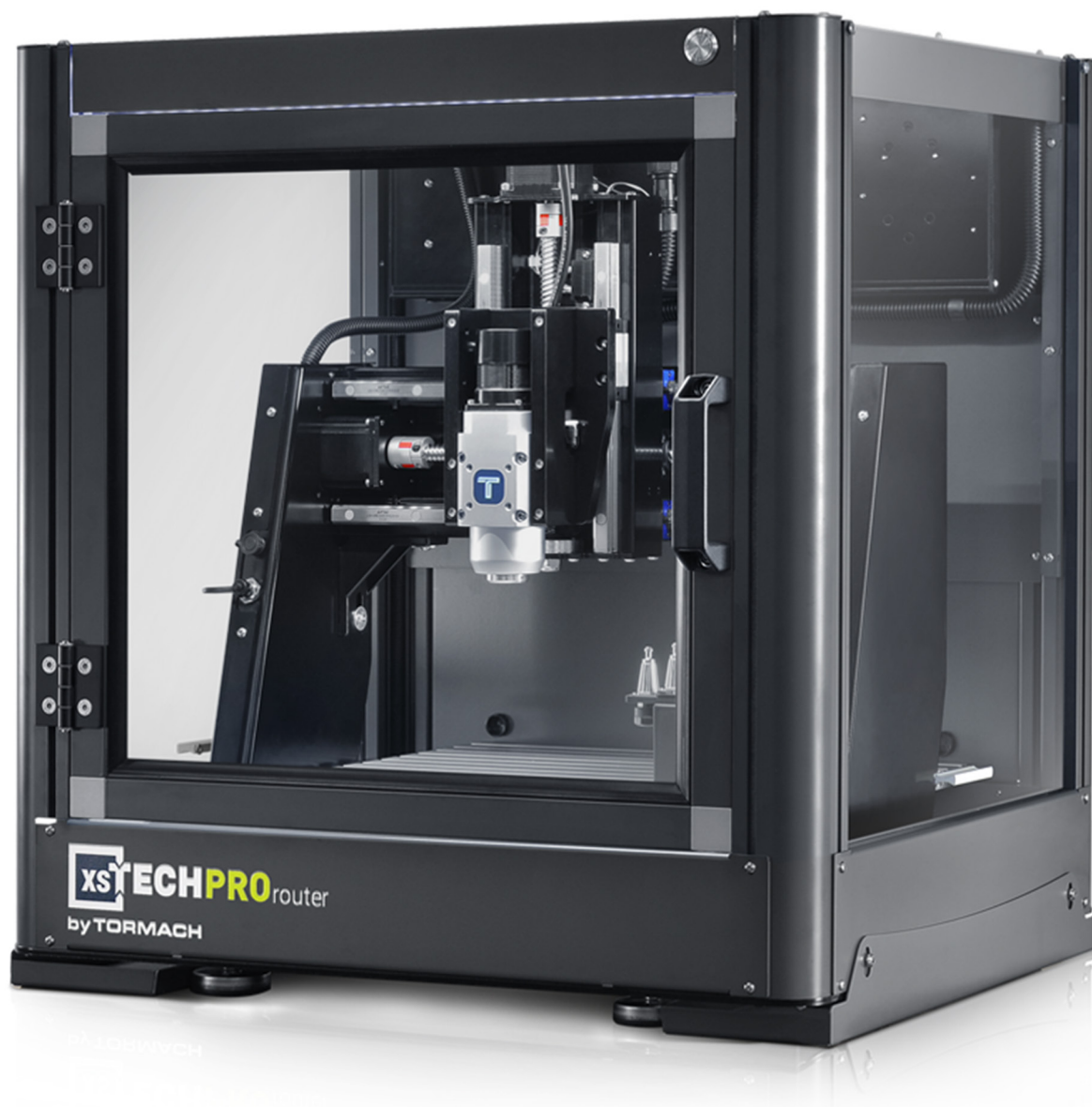




## Copyright

### Notice

Information is subject to change without notice by Tormach, Inc. For the most recent version of this document, see [tormach.com/support](http://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®, xsTECH Pro-CE®, and PathPilot® are trademarks or registered trademarks of Tormach, Inc. Our milling machines and accessories are covered by one or more of the following U.S. Patents: 7,386,362; D606,568; D612,406; D621,859; and other patent(s) pending.

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

Copyright © Tormach, Inc. 2026

---

### Tormach, Inc.

4009 Felland Road, Ste. 120

Madison, WI 53718 USA

[tormach.com](http://tormach.com)

---

### Authorized Representative

F2 Labs Engineering Department

5 Rue Parmentier, RDC

92200 Neuilly-sur-Seine

France

---

### Original Instructions

## 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](https://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](https://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](https://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](https://tormach.com/how-to-submit-a-support-ticket) for guidance on how to proceed.

## DECLARATION OF CONFORMITY

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

## LIABILITY DISCLAIMER

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

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

## EU DECLARATION OF CONFORMITY

**Product(s):** CNC Engraving Machine

**Model(s):** xsTECH Pro-CE (115 Vac and 230 Vac)

**Serial Number(s):** All beginning with EA, EB, or EC

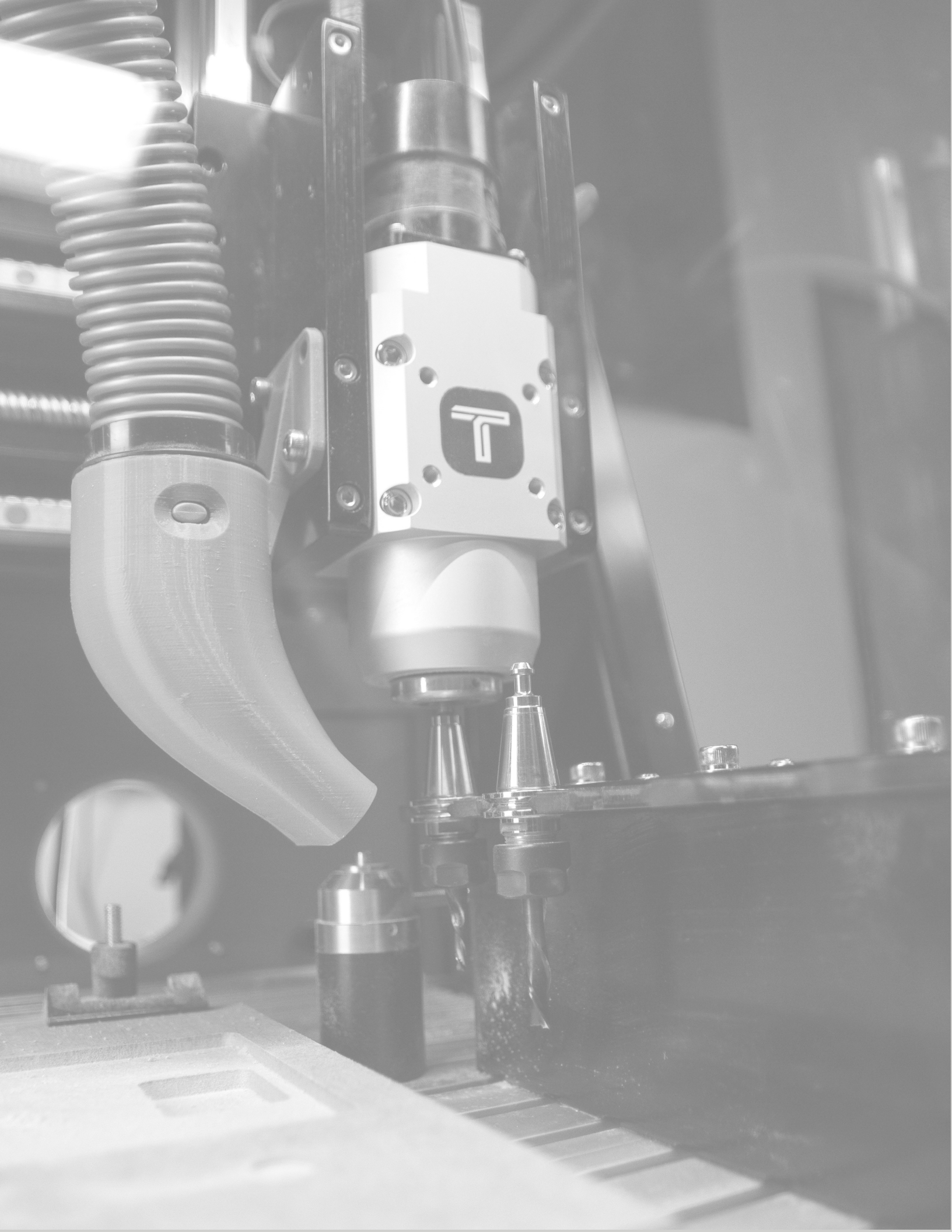
We, Tormach, Inc., hereby declare under our sole responsibility that the designated product(s) are in conformity with the following relevant European Union directive(s):

- **2006/42/EC** — Machinery Directive
- **2014/30/EU** — Electromagnetic Compatibility Directive

And are in conformity with the following standard(s) or normative document(s):

- **EN 60204-1:2010** Safety of Machinery — Electrical Equipment of Machines
- **ISO 12100:2010** Safety of Machinery — General Principles for Design — Risk Assessment and Risk Reduction
- **ISO 13849-1:2015** Safety-Related Parts of Control Systems — Part 1: General Principles for Design
- **EN 61000-6-2:2005** Electromagnetic Compatibility (EMC) — Part 6-2: Generic Standards — Immunity for Industrial Environments
- **EN 61000-6-4:2007+A1:2011** Electromagnetic Compatibility (EMC) — Part 6-4: Generic Standards — Emission Standard for Industrial Environments

We declare the technical file for this machine was compiled meeting the provisions of the listed directive(s) above and undertake to forward these to market monitoring authorities by request. This declaration is invalidated by any modification outside the scope of those intended by the manufacturer.



|                                                                 |           |
|-----------------------------------------------------------------|-----------|
| <b>IMPORTANT INFORMATION: PLEASE READ FIRST.....</b>            | <b>3</b>  |
| <b>SAFETY.....</b>                                              | <b>17</b> |
| 1.1 Intended Use .....                                          | 18        |
| 1.2 Machine Standards .....                                     | 19        |
| 1.2.1 American National Safety Institute (ANSI).....            | 19        |
| 1.2.2 Occupational Safety and Health Administration (OSHA)..... | 19        |
| 1.3 Safety Overview.....                                        | 20        |
| 1.3.1 Safety Messages.....                                      | 20        |
| <i>Personal Injury.....</i>                                     | 20        |
| <i>Property Damage.....</i>                                     | 20        |
| 1.3.2 Safety Decals.....                                        | 20        |
| <i>On the Electrical Cabinet Door.....</i>                      | 20        |
| <i>Next to the Main Disconnect.....</i>                         | 21        |
| <i>On the Spindle Nose.....</i>                                 | 21        |
| 1.3.3 Information Decals.....                                   | 21        |
| <i>Serial Number Plate.....</i>                                 | 21        |
| 1.4 Machine Safety.....                                         | 23        |
| 1.4.1 General Shop Safety.....                                  | 23        |
| 1.4.2 Operational Safety.....                                   | 23        |
| <i>General.....</i>                                             | 23        |
| <i>Tooling.....</i>                                             | 24        |
| <i>Workholding.....</i>                                         | 24        |
| 1.4.3 Electrical Safety.....                                    | 24        |
| 1.5 Noise Exposure.....                                         | 26        |
| <b>ABOUT YOUR MACHINE.....</b>                                  | <b>27</b> |
| 2.1 Machine Specifications.....                                 | 28        |
| <b>INSTALLATION.....</b>                                        | <b>29</b> |
| 3.1 Before You Begin.....                                       | 30        |
| 3.2 Installation Tools and Items.....                           | 31        |
| 3.3 Move the Pallet.....                                        | 32        |
| 3.4 Unpack the Machine Crate.....                               | 33        |
| 3.5 Install Core Components.....                                | 35        |
| 3.5.1 Unpack the Accessory Boxes.....                           | 35        |
| 3.5.2 Install the Controller.....                               | 35        |
| <i>Configure the Machine for 230 Vac Input.....</i>             | 37        |
| 3.5.3 Install the Pneumatic Kit.....                            | 38        |
| <i>Install the Pneumatic Module.....</i>                        | 38        |

# TABLE OF CONTENTS

|                                                                            |                 |
|----------------------------------------------------------------------------|-----------------|
| <i>Make Air Connections</i> .....                                          | 38              |
| 3.5.4 Power On the Machine .....                                           | 39              |
| <i>BIOS Configuration</i> .....                                            | 40              |
| 3.5.5 Align the ATC Rack .....                                             | 40              |
| <b>3.6 Install Accessory Components</b> .....                              | <b>41</b>       |
| 3.6.1 Install the Dust Collection Kit .....                                | 41              |
| <i>(Optional) Install the AC Contactor Box for PathPilot Control</i> ..... | 42              |
| 3.6.2 Install the Probe .....                                              | 43              |
| <i>Load and Verify the Probe</i> .....                                     | 43              |
| <i>Set the Reference Height</i> .....                                      | 43              |
| <i>Set the Probe Tip Diameter</i> .....                                    | 44              |
| 3.6.3 Set up the Electronic Tool Setter (ETS) .....                        | 44              |
| <i>Verify Tool Setter Function</i> .....                                   | 44              |
| <i>Set the Tool Setter Reference Height</i> .....                          | 44              |
| <i>Set the G37 Reference Position</i> .....                                | 45              |
| <b>3.7 Set Up the PathPilot Controller</b> .....                           | <b>46</b>       |
| 3.7.1 Specify the Date and Time .....                                      | 46              |
| 3.7.2 Specify the Keyboard Language .....                                  | 46              |
| 3.7.3 Update PathPilot .....                                               | 46              |
| <hr/> <b>PATHPILOT INTERFACE OVERVIEW</b> .....                            | <hr/> <b>47</b> |
| <b>4.1 About PathPilot</b> .....                                           | <b>48</b>       |
| <b>4.2 Notebook Section</b> .....                                          | <b>49</b>       |
| 4.2.1 Main Tab .....                                                       | 49              |
| 4.2.2 File Tab .....                                                       | 49              |
| 4.2.3 Settings Tab .....                                                   | 49              |
| 4.2.4 Offsets Tab .....                                                    | 50              |
| 4.2.5 Conversational Tab .....                                             | 50              |
| 4.2.6 Probe/ETS Tab .....                                                  | 50              |
| 4.2.7 Status Tab .....                                                     | 50              |
| <b>4.3 Persistent Controls</b> .....                                       | <b>52</b>       |
| 4.3.1 Program Control Area .....                                           | 52              |
| 4.3.2 Position Status Area .....                                           | 52              |
| 4.3.3 Manual Control Area .....                                            | 52              |
| <b>4.4 Keyboard Shortcuts</b> .....                                        | <b>54</b>       |
| <b>4.5 Manage PathPilot Versions</b> .....                                 | <b>55</b>       |
| 4.5.1 Download and Install an Update File from the Controller .....        | 55              |
| 4.5.2 Install an Update File from a USB Drive .....                        | 55              |
| 4.5.3 Install a Previous Version of an Update File .....                   | 56              |
| <hr/> <b>PATHPILOT TOOLS AND FEATURES</b> .....                            | <hr/> <b>57</b> |
| <b>5.1 Create and Load G-Code Files</b> .....                              | <b>58</b>       |
| 5.1.1 Load G-Code .....                                                    | 58              |
| <i>Transfer Files to and From the Controller</i> .....                     | 58              |
| <i>Preview G-Code Files</i> .....                                          | 58              |

|                                                  |    |
|--------------------------------------------------|----|
| Access Recent G-Code Files.....                  | 59 |
| Close the Current Program.....                   | 59 |
| 5.1.2 Edit G-Code.....                           | 59 |
| Edit G-Code with a Text Editor.....              | 59 |
| Edit G-Code with Conversational Programming..... | 59 |
| Change the Step Order.....                       | 60 |
| Create a New Job Assignment.....                 | 60 |
| Load an Existing G-Code File.....                | 60 |
| Edit a Job Assignment.....                       | 60 |
| 5.1.3 Read G-Code.....                           | 60 |
| Expand the G-Code Tab.....                       | 60 |
| About the G-Code Tab.....                        | 60 |
| Search in the Code.....                          | 61 |
| Set a New Start Line.....                        | 61 |
| Lead-In Moves.....                               | 62 |
| Change the View of the Tool Path Display.....    | 62 |
| About the Tool Path Display.....                 | 63 |
| 5.1.4 Use Conversational Programming.....        | 63 |
| About Conversational Programming.....            | 63 |
| Create a Face on a Part.....                     | 63 |
| About Facing.....                                | 64 |
| Facing in PathPilot.....                         | 64 |
| Facing Reference.....                            | 65 |
| Create a Profile on a Part.....                  | 65 |
| About Profiling.....                             | 65 |
| Profiling in PathPilot.....                      | 65 |
| Profiling Reference.....                         | 66 |
| Create a Pocket on a Part.....                   | 66 |
| About Pockets.....                               | 67 |
| Creating Pockets in PathPilot.....               | 67 |
| About Circular Pockets.....                      | 67 |
| About Rectangular Pockets.....                   | 68 |
| Create Hole Locations on a Part (Drill).....     | 68 |
| About Drilling.....                              | 69 |
| Create a Drilling Sequence.....                  | 69 |
| Drilling Reference.....                          | 69 |
| Create Threads on a Part.....                    | 70 |
| About Thread Milling.....                        | 71 |
| Thread Milling in PathPilot.....                 | 71 |
| Thread Milling Reference.....                    | 71 |
| Create Text to Engrave on a Part.....            | 71 |
| About Engraving.....                             | 72 |
| Import a DXF File.....                           | 72 |
| Working with Layers and Shapes.....              | 73 |
| Change the Layer or Shape Cut Order.....         | 73 |

# TABLE OF CONTENTS

- Adjust the Tree View Window.*.....73
  - Working in the Preview Window.*.....73
- 5.2 Machine Settings and Accessories.....74**
  - 5.2.1 Enable an Internet Connection.....74
    - Enable Automatic Updates.*.....74
  - 5.2.2 Change the Network Name.....74
  - 5.2.3 Change the Screen Orientation.....75
    - About Portrait Screen Layout.*.....75
  - 5.2.4 Disable Limit Switches.....76
    - About Limit Switches.*.....77
  - 5.2.5 Limit G30 Moves.....77
    - About G30.*.....77
  - 5.2.6 Enable the On-Screen Keyboard.....77
    - About Soft Keyboards.*.....78
  - 5.2.7 Enable the USB M-Code I/O Interface Kit.....78
  - 5.2.8 Enable Tooltips.....78
  - 5.2.9 Enable Feeds and Speeds Suggestions in Conversational Routines.....78
  - 5.2.10 Use a USB Camera.....78
    - About USB Cameras.*.....79
    - Manual Recording.*.....79
    - Automatic E-Stop Loop Recording ("Dashcam").*.....80
    - Review Video and Image Files.*.....80
      - File Naming Convention.....80
    - G-Code Commands.*.....81
      - File Naming Conventions.....81
      - Use M01 to Take Pictures.....81
- 5.3 Set Up G-Code Programs.....82**
  - 5.3.1 Use a Probe with PathPilot.....82
    - Set Up the Probe.*.....82
    - Use a Probe to Find a Feature's Location.*.....82
    - Use a Probe to Set Work Offset Zeroes.*.....82
      - Set the X and Y Work Offset Zero on the Corner of a Feature.....82
      - Set the Work Offset Zeroes on a Feature.....82
    - Use a Probe to Find the Center of a Feature.*.....83
      - Find the Center of a Pocket.....83
      - Find the Center of a Slot.....83
      - Find the Center of a Rectangular Boss.....83
      - Find the Center of a Circular Boss.....83
      - Find the Center Rotation of an A-Axis.....84
  - 5.3.2 Set Tool Length Offsets.....84
    - About Tool Offsets.*.....84
    - Measure Tools Using an Electronic Tool Setter (ETS).*.....84
      - Manually Measure Tool Lengths (with PathPilot).....85
      - Automatically Measure Tool Lengths (with G37).....85
      - Set up the Electronic Tool Setter (ETS).....85

|                                                                           |           |
|---------------------------------------------------------------------------|-----------|
| <i>Set the ETS Height.</i>                                                | 86        |
| <i>Reference the Spindle Nose.</i>                                        | 86        |
| <i>Set the G37 Position.</i>                                              | 86        |
| 5.3.3 Set Work Offsets.                                                   | 86        |
| <i>Set the Z-Axis Work Offset with an Electronic Tool Setter (ETS).</i>   | 87        |
| <i>About Work Offsets.</i>                                                | 87        |
| 5.3.4 View Work Offsets.                                                  | 88        |
| 5.3.5 View Available G-Code Modes.                                        | 88        |
| <b>5.4 Run G-Code Programs.</b>                                           | <b>89</b> |
| 5.4.1 Bring the Machine Out of Reset.                                     | 89        |
| <i>About Reset Mode.</i>                                                  | 89        |
| 5.4.2 View the Active Axis to Jog.                                        | 89        |
| 5.4.3 Jog the Machine.                                                    | 89        |
| <i>Jog in Continuous Velocity Mode.</i>                                   | 90        |
| <i>About Continuous Velocity Jogging.</i>                                 | 90        |
| <i>Jog in Step Mode.</i>                                                  | 90        |
| <i>About Step Jogging.</i>                                                | 90        |
| 5.4.4 View the Current Machine Position.                                  | 90        |
| 5.4.5 Reference the Machine.                                              | 90        |
| <i>About Referencing.</i>                                                 | 91        |
| 5.4.6 Start a Program.                                                    | 91        |
| <i>About Cycle Start.</i>                                                 | 91        |
| <i>Cycle Start Reference.</i>                                             | 91        |
| 5.4.7 Stop Machine Motion.                                                | 91        |
| 5.4.8 Operate the Coolant Pump.                                           | 91        |
| <i>About Coolant.</i>                                                     | 92        |
| 5.4.9 View the Active G-Code Modes.                                       | 92        |
| 5.4.10 View the Distance to Go.                                           | 92        |
| <i>About Distance to Go.</i>                                              | 92        |
| <b>5.5 Control G-Code Programs.</b>                                       | <b>93</b> |
| 5.5.1 Use the Feed Hold Function.                                         | 93        |
| <i>About Feed Hold.</i>                                                   | 93        |
| 5.5.2 Use the Feed Rate Override Function.                                | 93        |
| <i>About Feed Rate Override.</i>                                          | 93        |
| 5.5.3 Use M01 Break Mode.                                                 | 94        |
| <i>About M01 Break.</i>                                                   | 94        |
| <i>Display Information and Capture Images During an M00 or M01 Break.</i> | 94        |
| Display Information with Images.                                          | 94        |
| Display Information with Text.                                            | 94        |
| Capture Images with a USB Camera.                                         | 94        |
| 5.5.4 Use the Maxvel Override Function.                                   | 94        |
| <i>About Maxvel Override.</i>                                             | 95        |
| 5.5.5 Use Single Block Mode.                                              | 95        |
| <i>About Single Block.</i>                                                | 95        |
| 5.5.6 Use the Spindle Override Function.                                  | 95        |

# TABLE OF CONTENTS

|                                                   |            |
|---------------------------------------------------|------------|
| <i>About Spindle Override</i> .....               | 95         |
| 5.5.7 Change the Feed Rate.....                   | 96         |
| <i>About Feed Rates</i> .....                     | 96         |
| 5.5.8 Change the Spindle Speed.....               | 96         |
| <i>About Spindle Controls</i> .....               | 96         |
| 5.5.9 Change the Tool Number.....                 | 96         |
| <i>About M6 G43</i> .....                         | 96         |
| 5.5.10 Use a G30 Position.....                    | 96         |
| <i>About G30</i> .....                            | 97         |
| 5.5.11 View the Tool Length.....                  | 97         |
| 5.5.12 Manually Enter Commands.....               | 97         |
| <i>About the MDI Line DRO Field</i> .....         | 97         |
| <i>Admin Commands Reference</i> .....             | 97         |
| 5.5.13 Copy Recently Entered Commands.....        | 98         |
| 5.5.14 Use Feeds and Speeds Suggestions.....      | 98         |
| <i>Refresh DRO Field Values</i> .....             | 99         |
| <i>Use Additional Provided Information</i> .....  | 99         |
| <i>Create Tool Descriptions</i> .....             | 100        |
| Manually Enter Tool Descriptions.....             | 100        |
| Automatically Generate Tool Descriptions.....     | 100        |
| <i>Tool Keywords Reference</i> .....              | 101        |
| 5.5.15 Use Cycle Counters (M30 and M99).....      | 101        |
| <i>Monitor Cycle Counters</i> .....               | 102        |
| <i>Change Cycle Counter Values</i> .....          | 102        |
| <b>5.6 System File Management</b> .....           | <b>103</b> |
| 5.6.1 Manage System Files.....                    | 103        |
| 5.6.2 Create Backup Files.....                    | 103        |
| <i>About Backup Files</i> .....                   | 104        |
| 5.6.3 Restore Backup Files.....                   | 104        |
| 5.6.4 Import and Export the Tool Table.....       | 105        |
| <i>Import a .csv File</i> .....                   | 105        |
| <i>Export the Tool Table as a .csv File</i> ..... | 105        |
| <b>BASIC OPERATIONS</b> .....                     | <b>107</b> |
| <b>6.1 Start the Machine</b> .....                | <b>108</b> |
| <b>6.2 Bring the Machine Out of Reset</b> .....   | <b>109</b> |
| 6.2.1 About Reset Mode.....                       | 109        |
| <b>6.3 Reference the Machine</b> .....            | <b>110</b> |
| 6.3.1 About Referencing.....                      | 110        |
| <b>6.4 Jog the Machine</b> .....                  | <b>111</b> |
| 6.4.1 About Jogging.....                          | 111        |
| <i>About Continuous Velocity Jogging</i> .....    | 111        |
| <i>About Step Jogging</i> .....                   | 111        |
| 6.4.2 Jog Controls Reference.....                 | 112        |
| <i>Jogging in PathPilot</i> .....                 | 112        |

|                                                                             |            |
|-----------------------------------------------------------------------------|------------|
| <i>Jogging with the Keyboard.</i>                                           | 112        |
| <i>Jogging with the (Optional) Jog Shuttle or Operator Console Pendant.</i> | 112        |
| <b>6.5 Manually Control the Spindle</b>                                     | <b>113</b> |
| 6.5.1 About the Spindle.                                                    | 113        |
| <b>6.6 Load G-Code</b>                                                      | <b>114</b> |
| <b>6.7 Set Tool Length Offsets</b>                                          | <b>115</b> |
| 6.7.1 About Tool Offsets.                                                   | 115        |
| <i>Set up the Electronic Tool Setter (ETS).</i>                             | 115        |
| Set the ETS Height.                                                         | 115        |
| Reference the Spindle Nose.                                                 | 115        |
| Set the G37 Position.                                                       | 116        |
| <i>Measure Tools Using an Electronic Tool Setter (ETS).</i>                 | 116        |
| Manually Measure Tool Lengths (with PathPilot).                             | 116        |
| Automatically Measure Tool Lengths (with G37).                              | 117        |
| <b>6.8 Set Work Offsets</b>                                                 | <b>118</b> |
| 6.8.1 Set the Z-Axis Work Offset with an Electronic Tool Setter (ETS).      | 118        |
| 6.8.2 About Work Offsets.                                                   | 118        |
| <b>6.9 Operate the Coolant Pump</b>                                         | <b>120</b> |
| 6.9.1 About Coolant                                                         | 120        |
| <b>PROGRAMMING</b>                                                          | <b>121</b> |
| <b>7.1 Before You Begin</b>                                                 | <b>122</b> |
| <b>7.2 Programming Overview</b>                                             | <b>123</b> |
| 7.2.1 About G-Code Programming Language.                                    | 123        |
| 7.2.2 G-Code Formatting Reference.                                          | 123        |
| <i>Line Numbers.</i>                                                        | 123        |
| <i>Words.</i>                                                               | 123        |
| <i>Letters.</i>                                                             | 124        |
| <i>Values.</i>                                                              | 124        |
| <i>Order of Execution.</i>                                                  | 125        |
| <i>Modal Groups.</i>                                                        | 126        |
| <i>Comments.</i>                                                            | 127        |
| 7.2.3 Supported G-Codes Reference.                                          | 127        |
| <b>7.3 Programming G-Code</b>                                               | <b>129</b> |
| 7.3.1 About the Examples Used                                               | 129        |
| 7.3.2 Rapid Linear Motion (G00).                                            | 129        |
| 7.3.3 Linear Motion at Feed Rate (G01).                                     | 129        |
| 7.3.4 Arc at Feed Rate (G02 and G03).                                       | 130        |
| <i>Radius Format Arc.</i>                                                   | 130        |
| <i>Center Format Arc.</i>                                                   | 130        |
| Arc in XY Plane.                                                            | 131        |
| Arc in XZ Plane.                                                            | 131        |
| Arc in YZ Plane.                                                            | 131        |
| 7.3.5 Dwell (G04).                                                          | 132        |
| 7.3.6 Set Offsets (G10).                                                    | 132        |

# TABLE OF CONTENTS

|                                                                                |            |
|--------------------------------------------------------------------------------|------------|
| <i>Set Tool Table (G10 L1)</i> .....                                           | 132        |
| <i>Set Coordinate System (G10 L2)</i> .....                                    | 132        |
| <i>Set Tool Table (G10 L10)</i> .....                                          | 132        |
| <i>Set Tool Table (G10 L11)</i> .....                                          | 133        |
| <i>Set Coordinate System (G10 L20)</i> .....                                   | 133        |
| 7.3.7 Plane Selection (G17, G18, G19).....                                     | 133        |
| 7.3.8 Length Units (G20 and G21).....                                          | 133        |
| 7.3.9 Return to Predefined Position (G28 and G28.1).....                       | 133        |
| 7.3.10 Return to Predefined Position (G30 and G30.1).....                      | 134        |
| 7.3.11 Automatically Measure Tool Lengths with an ETS (G37 and G37.1).....     | 134        |
| <i>Move to G37 Position Over ETS (G37.1)</i> .....                             | 134        |
| <i>Move and Measure Tool Length (G37)</i> .....                                | 134        |
| 7.3.12 Straight Probe (G38.x).....                                             | 135        |
| <i>Use the Straight Probe Command</i> .....                                    | 135        |
| 7.3.13 Cutter Compensation (G40, G41, G42).....                                | 136        |
| 7.3.14 Dynamic Cutter Compensation (G41.1 and G42.1).....                      | 136        |
| 7.3.15 Apply Tool Length Offset (G43).....                                     | 137        |
| 7.3.16 Engrave Sequential Serial Number (G47).....                             | 137        |
| 7.3.17 Cancel Tool Length Compensation (G49).....                              | 137        |
| 7.3.18 Absolute Coordinates (G53).....                                         | 137        |
| 7.3.19 Select Work Offset Coordinate System (G54 to G54.1 P500).....           | 138        |
| 7.3.20 Set Exact Path Control Mode (G61).....                                  | 138        |
| 7.3.21 Set Blended Path Control Mode (G64).....                                | 138        |
| 7.3.22 Distance Mode (G90 and G91).....                                        | 138        |
| 7.3.23 Arc Distance Mode (G90.1 and G91.1).....                                | 138        |
| 7.3.24 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3).....              | 138        |
| 7.3.25 Feed Rate Mode (G93, G94, and G95).....                                 | 139        |
| 7.3.26 Spindle Control Mode (G96 and G97).....                                 | 139        |
| <b>7.4 Programming Canned Cycles</b> .....                                     | <b>141</b> |
| 7.4.1 Canned Cycles Reference.....                                             | 141        |
| <i>Supported Canned Cycles</i> .....                                           | 141        |
| 7.4.2 High Speed Peck Drill (G73).....                                         | 142        |
| 7.4.3 Cancel Canned Cycles (G80).....                                          | 142        |
| 7.4.4 Drilling Cycle (G81).....                                                | 142        |
| 7.4.5 Simple Drilling Cycle (G82).....                                         | 144        |
| 7.4.6 Peck Drilling Cycle (G83).....                                           | 144        |
| 7.4.7 Boring Cycle (G85).....                                                  | 144        |
| 7.4.8 Boring Cycle (G86).....                                                  | 144        |
| 7.4.9 Boring Cycle (G88).....                                                  | 144        |
| 7.4.10 Boring Cycle (G89).....                                                 | 144        |
| <b>7.5 Programming M-Code</b> .....                                            | <b>146</b> |
| 7.5.1 Supported M-Codes Reference.....                                         | 146        |
| 7.5.2 Program Stop and Program End (M00, M01, M02, and M30).....               | 146        |
| <i>Display Information and Capture Images During an M00 or M01 Break</i> ..... | 147        |
| Display Information with Images.....                                           | 147        |

|                                                    |            |
|----------------------------------------------------|------------|
| Display Information with Text .....                | 147        |
| Capture Images with a USB Camera .....             | 147        |
| 7.5.3 Spindle Control (M03 and M05) .....          | 147        |
| 7.5.4 Tool Change (M06) .....                      | 147        |
| 7.5.5 Coolant Control (M08 and M09) .....          | 148        |
| 7.5.6 Override Control (M48 and M49) .....         | 148        |
| 7.5.7 Feed Override Control (M50) .....            | 148        |
| 7.5.8 Spindle Speed Override Control (M51) .....   | 148        |
| 7.5.9 Set Current Tool Number (M61) .....          | 148        |
| 7.5.10 Set Output State (M64 and M65) .....        | 148        |
| 7.5.11 Wait on Input (M66) .....                   | 148        |
| 7.5.12 USB Camera Control (M301, M302, M303) ..... | 149        |
| <b>7.6 Programming Input Codes .....</b>           | <b>150</b> |
| 7.6.1 Feed Rate (F) .....                          | 150        |
| 7.6.2 Spindle Speed (S) .....                      | 150        |
| 7.6.3 Change Tool Number (T) .....                 | 150        |
| <b>7.7 Advanced Programming .....</b>              | <b>151</b> |
| 7.7.1 Parameters .....                             | 151        |
| <i>Parameters Reference</i> .....                  | 151        |
| Parameter Syntax .....                             | 151        |
| Parameter Scope .....                              | 151        |
| Behavior of Uninitialized Parameters .....         | 151        |
| Parameter Mode .....                               | 151        |
| Persistence and Volatility .....                   | 151        |
| Intended Use .....                                 | 151        |
| <i>Numbered Parameters Reference</i> .....         | 152        |
| Read-Only Parameters .....                         | 152        |
| <i>Subroutine Parameters Reference</i> .....       | 152        |
| <i>Named Parameters Reference</i> .....            | 152        |
| 7.7.2 Expressions .....                            | 153        |
| <i>Binary Operators Reference</i> .....            | 153        |
| <i>Functions Reference</i> .....                   | 153        |
| 7.7.3 Subroutines .....                            | 154        |
| <i>Subroutines Reference</i> .....                 | 154        |
| <i>Conditional Subroutines Reference</i> .....     | 155        |
| if/endif .....                                     | 155        |
| if/elseif/else/endif .....                         | 155        |
| <i>Repeating Subroutines Reference</i> .....       | 155        |
| <i>Looping Subroutines Reference</i> .....         | 155        |
| do/while .....                                     | 156        |
| while/endwhile .....                               | 156        |
| <b>MACHINE MAINTENANCE .....</b>                   | <b>157</b> |
| <b>8.1 Maintenance Safety .....</b>                | <b>158</b> |
| 8.1.1 All Maintenance Procedures .....             | 158        |

# TABLE OF CONTENTS

|                                                |            |
|------------------------------------------------|------------|
| 8.1.2 Swarf Maintenance Procedures.....        | 158        |
| <b>8.2 Maintenance Schedules.....</b>          | <b>159</b> |
| 8.2.1 Daily.....                               | 159        |
| 8.2.2 Monthly (or every ~80 hours).....        | 159        |
| 8.2.3 Semi-Annually (or Every ~250 hours)..... | 159        |
| 8.2.4 Annually (or as needed).....             | 159        |
| <hr/>                                          |            |
| <b>DIAGRAMS AND PARTS LISTS.....</b>           | <b>161</b> |
| <hr/>                                          |            |
| <b>9.1 xsTech Pro.....</b>                     | <b>162</b> |
| 9.1.1 Front View.....                          | 162        |
| 9.1.2 Rear View.....                           | 163        |
| 9.1.3 Machine View (Enclosure Removed).....    | 164        |
| <hr/>                                          |            |
| <b>ELECTRICAL SCHEMATICS.....</b>              | <b>167</b> |
| <hr/>                                          |            |
| 10.1 Power (Page 2).....                       | 168        |
| 10.2 Switches (Page 3).....                    | 169        |
| 10.3 Spindle (Page 4).....                     | 170        |
| 10.4 Axis Motors (Page 5).....                 | 171        |
| 10.5 Ethernet Cable (Page 6).....              | 172        |
| 10.6 Solenoids and Relays (Page 7).....        | 173        |
| 10.7 USB Hub (Page 8).....                     | 174        |

# 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

|                                    |           |
|------------------------------------|-----------|
| <b>1.1 Intended Use</b> .....      | <b>18</b> |
| <b>1.2 Machine Standards</b> ..... | <b>19</b> |
| <b>1.3 Safety Overview</b> .....   | <b>20</b> |
| <b>1.4 Machine Safety</b> .....    | <b>23</b> |
| <b>1.5 Noise Exposure</b> .....    | <b>26</b> |

# 1: SAFETY

## 1.1 Intended Use

### 1.1 INTENDED USE

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

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

The intended use includes:

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

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

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

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

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

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

### 1.2 MACHINE STANDARDS

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

#### 1.2.1 American National Safety Institute (ANSI)

- **ANSI B11.TR3-2000** *Risk Assessment and Risk Reduction — A Guideline to Estimate, Evaluate, and Reduce Risks Associated with Machine Tools*

#### 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.

#### 1.3.1 Safety Messages

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

##### Personal Injury

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

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

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

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

##### Property Damage

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

#### 1.3.2 Safety Decals

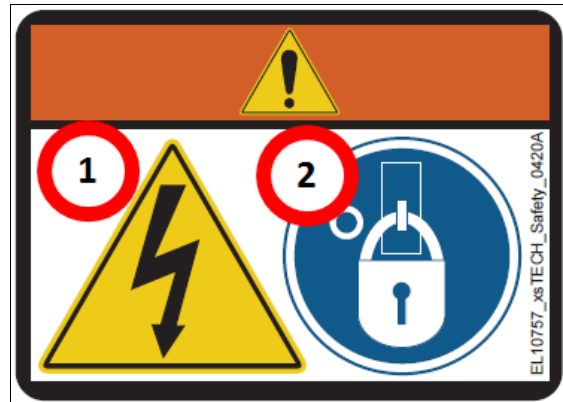
Before operating the machine in any way, you must read and

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

The following types of safety symbols are on the decals:

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

##### On the Electrical Cabinet Door



**Figure 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.

### Next to the Main Disconnect



**Figure 1-2: Example of a safety decal next to the Main Disconnect.**

1. **WARNING! Entanglement / Entrapment Hazard.** The machine operates under automatic control — it can start at any time and crush, cut, entangle, or pinch body parts. Always keep clear of positions on the machine where unexpected or unintended machine motion could cause harm. Before operating this machine in any way, you must verify that all operators know the location of the machine's Emergency Stop button.
2. **WARNING! Ejection Hazard.** Fixtures, tooling, workpieces, or other loose items can become dangerous projectiles and can cause death or serious injury. Before operating this machine in any way, you must verify that you have appropriately secured all components.
3. **WARNING! Fire Hazard.** 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.** 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 787+.

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.

### On the Spindle Nose



**Figure 1-3: Example of a safety decal on the spindle nose.**

1. **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.
2. **WARNING! Cut Hazard.** Tools and swarf can cut you. Only hold tools by the tool holder. Before inserting or removing tools from the machine, you must verify that all motion is completely stopped.

### 1.3.3 Information Decals

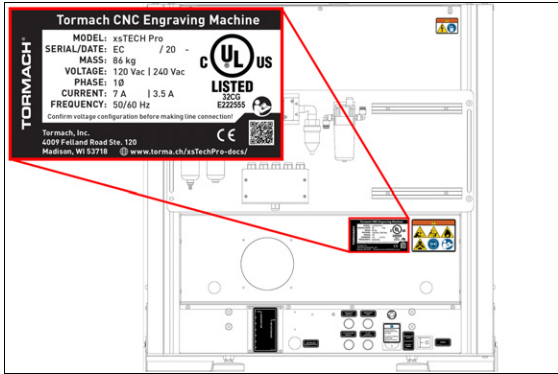
Before operating the machine in any way, you must locate and become familiar with all installed information decals on the machine and equipment.

#### Serial Number Plate

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

# 1: SAFETY

## 1.3 Safety Overview



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

### 1.4 MACHINE SAFETY



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

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

|                                  |           |
|----------------------------------|-----------|
| <b>1.4.1 General Shop Safety</b> | <b>23</b> |
| <b>1.4.2 Operational Safety</b>  | <b>23</b> |
| <b>1.4.3 Electrical Safety</b>   | <b>24</b> |

#### 1.4.1 General Shop Safety

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

#### 1.4.2 Operational Safety

##### General

- ✓ Understand that the machine is automatically controlled and can start at any time.
- ✓ Become familiar with all physical and software controls.
- ✓ Always use a chip scraper or brush when clearing away chips, oil, or coolant.
- ✓ Examine all tools, fixtures, workpieces, and guarding for signs of damage. Replace any damaged components as soon as you find them.  
The enclosure and other guards may not stop all types of projectiles, like broken tools or loose workpieces.
- ✓ Stop the machine and verify that all machine motion has completely stopped before doing any of the following:
  - ✓ Adjusting a part, fixture, or coolant nozzle.
  - ✓ Changing tools or parts.
  - ✓ Clearing away chips, oil, or coolant.
  - ✓ Reaching into any part of the machine's motion envelope.
  - ✓ Removing protective shields or safeguards.
  - ✓ Taking measurements.
  - ✓ Doing any other action inside the machine enclosure.
- ✓ Use flood or MQL (mist) coolant as required by the machining operation.
- ✓ Only use coolants designed for metal working applications such as soluble oils, semi-synthetic, or synthetic coolants.
- ✓ Read the Safety Data Sheet (SDS) for all workpiece materials, coatings, coolants (flood or MQL), lubricants, and other consumables. Chips, dust, and vapors from certain materials can be toxic or otherwise harmful.
- ✓ Understand that this product can expose you to chemicals which are known to the State of California to cause cancer and birth defects or other reproductive harm. For more information, go to [www.P65Warnings.ca.gov](http://www.P65Warnings.ca.gov).
- ✓ Dispose of scrap and swarf according to local regulations and guidelines.
- ✓ Thoroughly read all safety precautions and instructions.

# 1: SAFETY

## 1.4 Machine Safety

- ✓ When machining an unproven program, use feed, speed, and maximum velocity overrides, Distance-to-Go (DTG) displays, single block, feed hold, and other control features.
- ✓ Follow all appropriate "Machine Standards" (page 19).
- ✗ Never reach around a guard.
- ✗ Never allow the machine to run unattended.
- ✗ Never obstruct the Emergency Stop button or any other controls.
- ✗ Never allow untrained operators to install, operate, or maintain the machine.
- ✗ Never modify, defeat, or bypass safety devices or interlocks.
- ✗ Never machine abrasive, carcinogenic, explosive, flammable, radioactive, or toxic materials. Such materials include, but are not limited to:
  - ✗ Beryllium and its alloys
  - ✗ Ceramic
  - ✗ Fiberglass
  - ✗ G10 fiberglass laminate
  - ✗ Graphite
  - ✗ Lead and its alloys
  - ✗ Magnesium and its alloys
- ✗ Never allow swarf to accumulate on or within the machine.
- ✗ Never use flammable liquids (like alcohol, diesel fuel, or kerosene) in the machine's coolant system.
- ✗ Never use water, coolants without rust inhibitors, or straight cutting oil in the machine's coolant system.

### Tooling

- ✓ Use appropriate speeds, feeds, and cutting parameters for your machine, machine operation, material, and tooling.
- ✓ Use tools and tool holders that are suitable for the current operation.
- ✓ Examine tools for signs of damage. Replace any damaged tools as soon as you find them.  
The enclosure and other guards may not stop all types of projectiles, like broken tooling.
- ✗ Never use unbalanced tooling or spindle fixtures.
- ✗ Never use tools that are larger or longer than necessary.

- ✗ Never use tools at speeds above their operational limits.

### Workholding

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

### 1.4.3 Electrical Safety



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

- ✓ Power off the machine before servicing.
- ✓ 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.
- ✓ Confirm that the machine installation meets all codes and regulations of your locality.
- ✓ Confirm that electrical connections are performed by a certified electrician.
- ✓ Lock the electrical cabinet door and remove the keys when the machine is not being serviced to prevent unqualified or unauthorized personnel from accessing the electrical cabinet.
- ✗ Never operate the machine with the electrical cabinet door open.

- ✗ Never reach into the electrical cabinet with the machine powered on.
- ✗ Never modify the machine's electronics.
- ✗ Never drill into the electrical cabinet.

# 1: SAFETY

## 1.5 Noise Exposure

### 1.5 NOISE EXPOSURE



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

With the axes moving and the spindle rotating (but not cutting), the emission sound pressure level at the operator control station is <66 dB(A).

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

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

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

# ABOUT YOUR MACHINE

## IN THIS SECTION, YOU'LL LEARN:

- About this machine's specifications.

---

### CONTENTS

|                                 |    |
|---------------------------------|----|
| 2.1 Machine Specifications..... | 28 |
|---------------------------------|----|

## 2: ABOUT YOUR MACHINE

### 2.1 Machine Specifications

#### 2.1 MACHINE SPECIFICATIONS

| Machine Specifications   |                                                                                                                                                                                                   |
|--------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Work Area (X × Y × Z)    | 12" × 9" × 6.5" (305 mm × 229 mm × 165 mm)                                                                                                                                                        |
| Typical System Footprint | 22.5" × 23.6" × 26.4" (570 mm × 600 mm × 670 mm)                                                                                                                                                  |
| Machine Weight           | 150 lb (68 kg)                                                                                                                                                                                    |
| Power and Air            |                                                                                                                                                                                                   |
| Power Required           | USA, Mexico, and Canada: <ul style="list-style-type: none"><li>• 120 Vac</li><li>• 15 A breaker</li></ul> Europe: <ul style="list-style-type: none"><li>• 230 Vac</li><li>• 6 A breaker</li></ul> |
| Air Required             | 90-120 psi (620-825 kPa)                                                                                                                                                                          |
| Spindle                  |                                                                                                                                                                                                   |
| Spindle Power            | 350 W                                                                                                                                                                                             |
| Maximum Spindle Speed    | 20,000 rpm                                                                                                                                                                                        |
| Travels                  |                                                                                                                                                                                                   |
| X-Axis                   | 10.2" (260 mm)                                                                                                                                                                                    |
| Y-Axis                   | 7.9" (200 mm)                                                                                                                                                                                     |
| Z-Axis                   | 6.5" (160 mm)                                                                                                                                                                                     |

# 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

|                                           |    |
|-------------------------------------------|----|
| 3.1 Before You Begin .....                | 30 |
| 3.2 Installation Tools and Items .....    | 31 |
| 3.3 Move the Pallet .....                 | 32 |
| 3.4 Unpack the Machine Crate .....        | 33 |
| 3.5 Install Core Components .....         | 35 |
| 3.6 Install Accessory Components .....    | 41 |
| 3.7 Set Up the PathPilot Controller ..... | 46 |



**TORMACH®**

# 3: INSTALLATION

## 3.1 Before You Begin

### 3.1 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](https://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.

### 3.2 INSTALLATION TOOLS AND ITEMS

- Assistant to help you
- Cable ties, 8 in.
- Level
- Metric hex wrench set
- Pallet jack
- Phillips screwdriver
- Safety eyewear that meets ANSI Z87+
- Shears or knife
- Standard-depth countertop (24 in.) or similar, 36 in. minimum width
- Work gloves
- Work surface

## 3: INSTALLATION

### 3.3 Move the Pallet

#### 3.3 MOVE THE PALLET



**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).

#### 3.4 UNPACK THE MACHINE CRATE

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

1. Put on work gloves and eye protection.
2. Remove the plastic wrap around the outside of the crate.
3. Use a Phillips screwdriver to remove the screws from the top of the crate.



**Figure 3-1: Removing the screws on the top of the crate.**

4. Lift the top off the crate.
5. Remove the cardboard and other packaging from the top of the crate's interior.
6. Use a Phillips screwdriver to remove the corner brackets from sides of the crate.



**Figure 3-2: Removing the corner brackets from the sides of the crate.**

7. Remove the screws from each side of the crate, then remove all four side panels.

8. Carefully remove the plastic wrap and packaging from the machine. Do not use shears or a knife near the machine.



**Figure 3-3: Plastic wrap on the machine.**

9. Slide the two chip trays out from the front of the machine and set them aside.



**Figure 3-4: Removing the chip tray.**

10. Remove the lock screws on the machine's side window with the provided T-20 T-handle driver.
11. Open the side window and the front door, and remove the internal packaging. Remove the three accessory boxes from the machine table and set them aside. Close the side window.

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

## 3: INSTALLATION

### 3.4 Unpack the Machine Crate

12. Before you lift the machine, verify that you have assistance to help you move it off of the pallet. We recommend two people. Lift the machine and place it on the table.



**Figure 3-5: Lifting and placing the machine on the table.**

13. Check that the machine and table are level. If needed, adjust the jam nuts on the leveling feet.



**Figure 3-6: Jam nuts on the machine feet.**

14. Slide the chip trays back into their tracks on the front of the machine.

### 3.5 INSTALL CORE COMPONENTS

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

|                                              |           |
|----------------------------------------------|-----------|
| <b>3.5.1 Unpack the Accessory Boxes.....</b> | <b>35</b> |
| <b>3.5.2 Install the Controller.....</b>     | <b>35</b> |
| <b>3.5.3 Install the Pneumatic Kit.....</b>  | <b>38</b> |
| <b>3.5.4 Power On the Machine.....</b>       | <b>39</b> |
| <b>3.5.5 Align the ATC Rack.....</b>         | <b>40</b> |

#### 3.5.1 Unpack the Accessory Boxes

➤ Find the three accessory boxes that you set aside earlier, and open all three. The accessory boxes contain the following tools and items:

- **Box 1**
  - Pneumatic module with fasteners
  - ATC air line
  - Pneumatic air line
  - Air tool oil
- **Box 2**
  - Jog shuttle
  - Keyboard
  - Mouse
  - Emergency stop
  - Ethernet cable
  - USB-A to micro USB cable
  - Grease gun and grease
  - IEC cable (for machine to controller)
  - Basic workholding kit
  - Tools
  - Controller arm
  - Panel vise
  - Basic toolholder kit
  - Spare fuses
- **Box 3**
  - Controller
  - 120 Vac power cable
  - Cable split loom
  - Controller power supply
  - Cable loom clamps

- USB Wifi Module
- Antennas for onboard wifi (not currently used)
- Cable ties

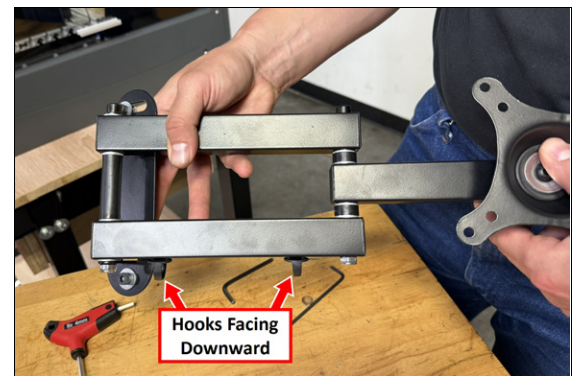
#### 3.5.2 Install the Controller

1. Open the controller arm box and locate the included hardware. Then, locate the wrench kit inside the accessory box.
2. Install the mounting hardware on the monitor arm by inserting a bolt and washer through both sides of the bracket, and threading each bolt into a T-nut from the underside.



**Figure 3-7: Installing the hardware onto the controller arm.**

3. Orient the monitor arm so that the hooks face downward.

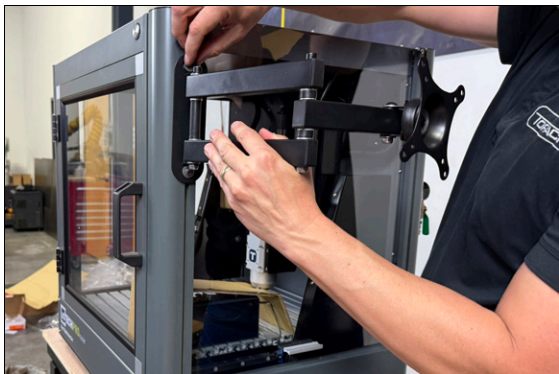


**Figure 3-8: Monitor arm with the hooks facing downward.**

## 3: INSTALLATION

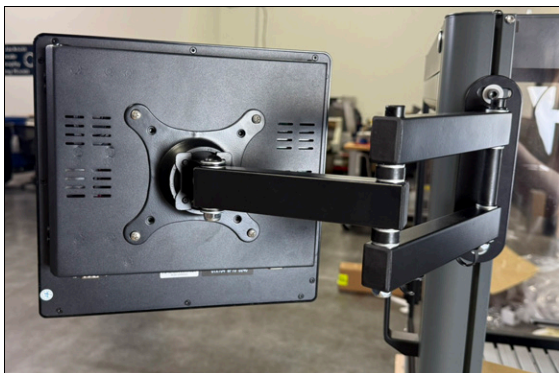
### 3.5 Install Core Components

- Slide the monitor arm into the slot until it's correctly positioned, and use a 4 mm hex wrench to tighten and remove any twist in the arm. To adjust the height, reposition the arm as needed.



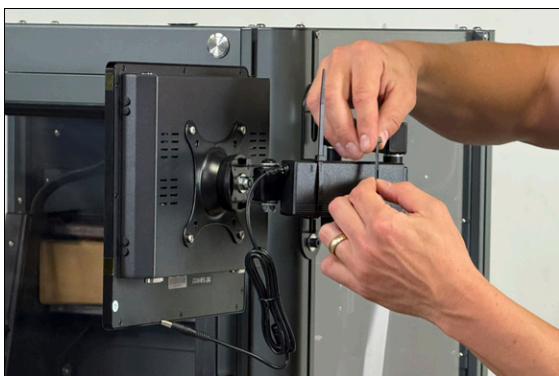
**Figure 3-9: Installing the controller arm onto the machine.**

- Locate the controller and its included cables. Mount the controller to the arm using the four included M4 screws from the controller hardware bag.



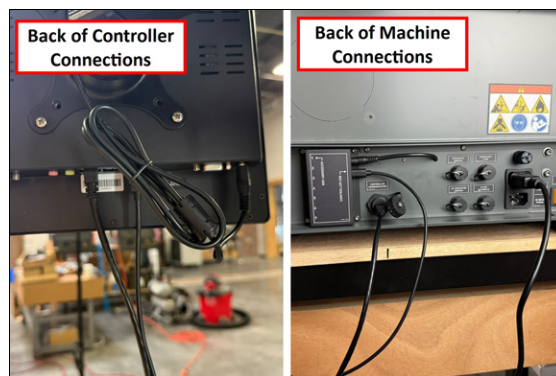
**Figure 3-10: Controller mounted to the arm.**

- Mount the power brick to the arm using two zip ties.



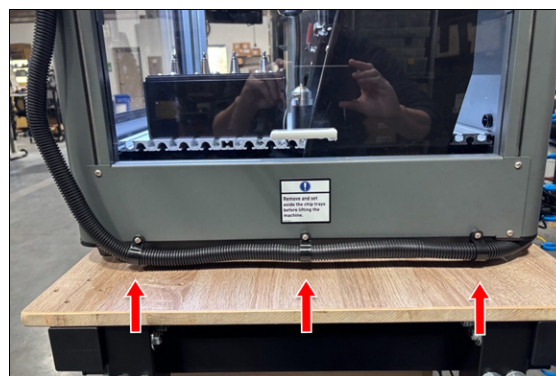
**Figure 3-11: Power brick mounted to the arm.**

- Make the following connections:
  - Power Cable** Connect to the controller.
  - Power Brick Extension** Connect one end to the power brick and route the loose end to the **Controller Power** port on the back of the machine.
  - Ethernet Cable** Connect one end to the controller and route the loose end to the **Controller Communications** port on the back of the machine.
  - USB Extension Cable** Connect one end to any USB port on the controller and route the loose end to the **Controller USB** port on the back of the machine.



**Figure 3-12: Controller connections.**

- Use cable ties to bundle the controller cables.
- Use the included split-loom cable carrier to secure all controller cables. Attach it to the side of the machine with the provided cable straps.



**Figure 3-13: Split-loom cable carrier.**

- Connect accessories to the controller:
  - WiFi Dongle** Insert into any available USB port on the back of the machine.
  - Jog Shuttle, Keyboard, and Mouse** Connect the USB cables to the available **Accessory USB** ports on the back of the machine.

11. Connect the cable from the emergency stop box to the **Emergency Stop** port on the back of the machine and place the box on the left side of the machine.
12. Connect the input power cable to the **Input Power** port on the back of machine.

**NOTICE!** This machine is configured for 120 Vac input. If you're installing the machine with 230 Vac power, you **must** configure the machine **before** applying power. Go to "Configure the Machine for 230 Vac Input" (below).

13. Plug the loose end of the input power cable into a wall outlet.

#### Configure the Machine for 230 Vac Input

The machine is configured for 120 Vac input. If your mains line voltage is 230 Vac, you must configure the machine as detailed in this procedure **before** connecting the machine to power.

**NOTICE!** Incorrect Voltage Configuration: You must complete the voltage conversion procedure before powering on the machine. If you don't, there's a risk of machine damage.



**Note:** There's no performance advantage to using 230 Vac. Only follow this procedure if it's required for your location.

1. Identify the 3.5 A fuse that's included with this machine (it's in the same box as the tool holders and collets).
2. On the back panel of the machine, locate the fuse holder. Press the fuse holder inward firmly (it is spring-loaded and may be snug), then twist counterclockwise to release it.



**Figure 3-14: Fuse holder on the back panel of the machine.**

3. Remove the 7.0 A fuse and replace it with the 3.5 A fuse.
4. Push the fuse holder back into position and twist clockwise to lock it in place.
5. Open the front door of the machine, and gently pull the gantry forward toward the front of the machine.
6. Open the left side window and, on the bottom of the electrical box, identify the dust cap. Remove the dust cap.



**Figure 3-15: Removing the dust cap.**

7. Flip the primary power supply's input voltage switch to the left with a flat-blade screwdriver. This switches the input from 120 Vac to 230 Vac. Replace the dust cap and close the window.
8. Replace the input power cord with the appropriate cord for your region (replacement cord is not provided). Use an IEC 60320 C13 cord (commonly used in many home electronics).

# 3: INSTALLATION

## 3.5 Install Core Components

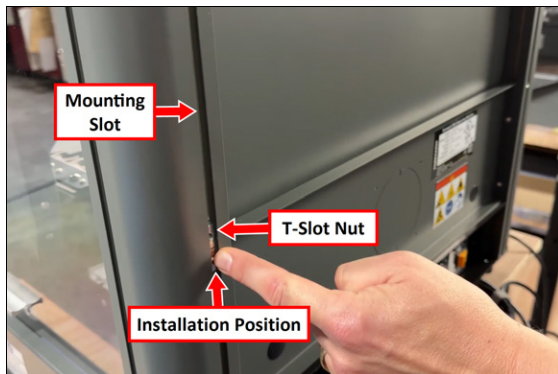
### 3.5.3 Install the Pneumatic Kit

#### Install the Pneumatic Module



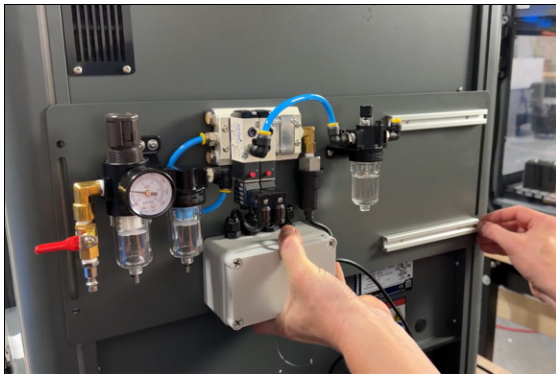
**Tip!** This procedure may be easier with assistance, though it can be completed alone.

1. Identify the four T-slot nuts and four M6 screws provided with the pneumatic kit. Then, slide a T-slot nut into the mounting slot on the left rear of the machine. Allow the nut to slide down until it stops at the installation position.



**Figure 3-16:** T-slot nut slid into the left mounting slot.

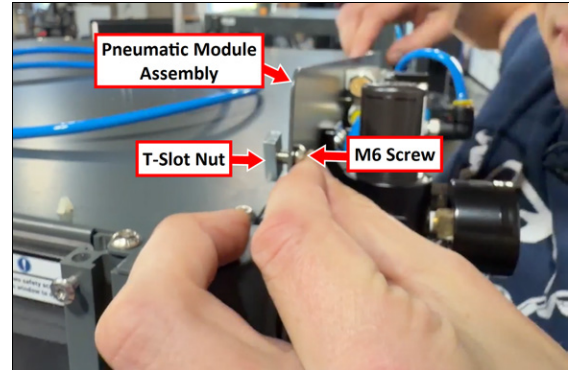
2. Repeat Step 1 to install a nut into the right rear mounting slot.
3. Lift the pneumatic module into position, aligning the lower mounting holes with the installed T-slot nuts.



**Figure 3-17:** Positioning the pneumatic module onto the back of the machine.

4. Thread two M6 screws through the mounting holes into the T-slot nuts and tighten them finger-tight (do not fully tighten yet).
5. Slide the entire pneumatic assembly upward until it's above the machine, exposing the upper mounting holes.

6. While holding the assembly in the raised position (or having someone hold it), thread an M6 screw through each upper mounting hole from the front while positioning a T-slot nut behind the mounting plate from the back.



**Figure 3-18:** Installing the hardware into the upper two mounting holes.

7. Rotate each T-slot nut until it aligns with its mounting slot, then guide the nut into the slot.
8. Slide the assembly back down until it stops at the installation position.
9. Tighten all four M6 screws to secure the pneumatic module to the machine.
10. Connect the pneumatic module's voltage cable to the **Pneumatic Module** connector on the back of the machine.

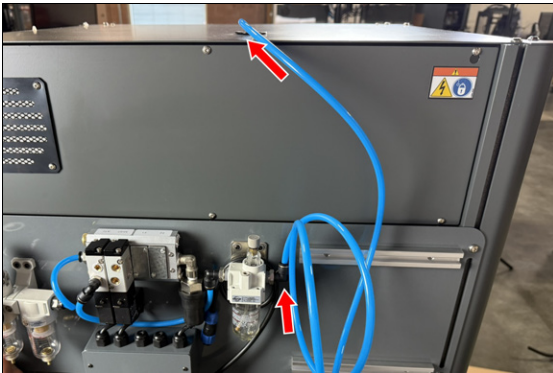


**Figure 3-19:** Pneumatic Module connector.

#### Make Air Connections

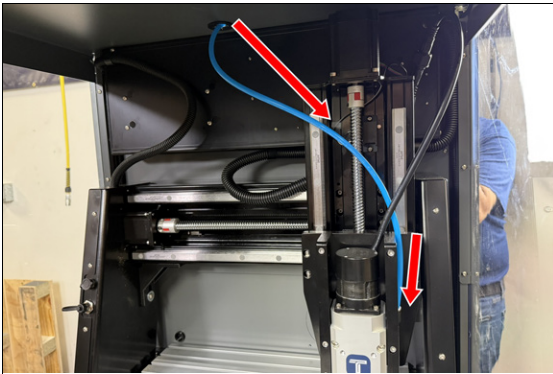
1. Remove any remaining packaging from inside the machine. Gently lift the Z-axis by hand to remove the packaging between the spindle and the ATC rack.

2. Locate the short blue airline (for the ATC):
  - a. Insert one end into the FRL unit on the pneumatic module (push-to-connect fitting).



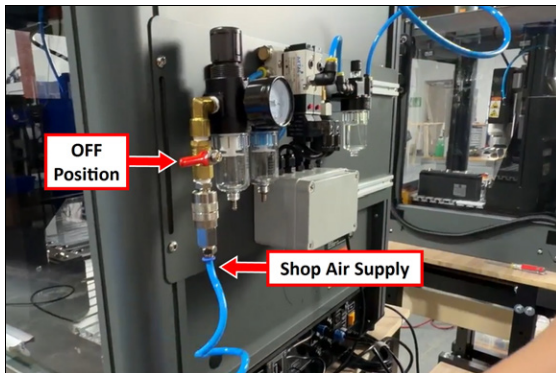
**Figure 3-20: Routing the short blue airline to the FRL.**

- b. Route the loose end through the hole on top of the enclosure, and connect it to the fitting on the side of the spindle.



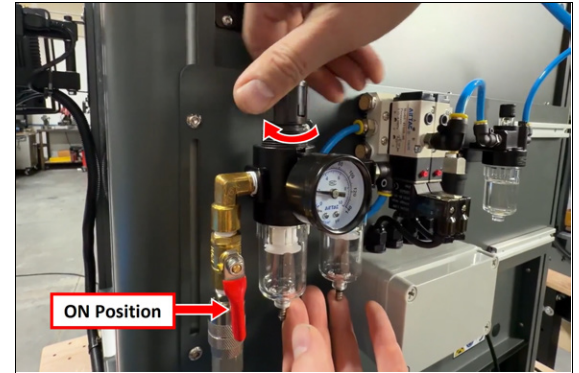
**Figure 3-21: Routing the short blue airline to the spindle.**

3. Connect the coiled shop air supply hose to the inlet port on the FRL unit on the pneumatic module.



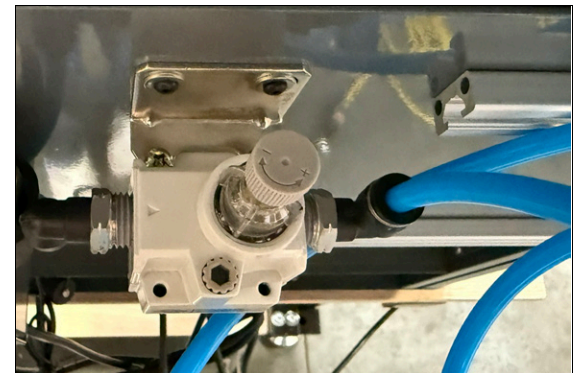
**Figure 3-22: Shop air connected to the inlet port of the FRL.**

4. Connect the loose end of the coiled hose into your shop's airline.
5. Open the valve on the FRL, and adjust the regulator by rotating the adjustment knob until the pressure gauge reads 90 psi (0.6 MPa), then push the knob down to lock it in place.



**Figure 3-23: Regulator adjusted to 90 psi.**

6. Close the valve on the FRL to depressurize the system.
7. Remove the lubricator fill screw on the FRL unit, fill the reservoir to approximately 50% capacity using the included air tool oil, then replace and tighten the fill screw.
8. Turn the lubricator adjustment valve fully clockwise to close it completely, then turn it counterclockwise **very** slightly to set the correct lubrication rate.



**Figure 3-24: FRL metering knob.**

#### 3.5.4 Power On the Machine

- Power on the machine and the PathPilot controller.
  1. Flip the Power switch to **ON** on the back of the machine.

## 3: INSTALLATION

### 3.5 Install Core Components

- Twist out the machine's red Emergency Stop button, which enables movement to the machine axes and the spindle.



**Note:** If the controller doesn't power on immediately when the main machine powers on, you may need to configure the BIOS settings. Follow the procedure detailed in "BIOS Configuration" (below).

#### BIOS Configuration

- Press the Power button on the bottom left of the controller.
- While the controller boots, quickly press and hold the **ESC** key on the keyboard as the American Megatrends bootscreen appears to enter the BIOS configuration.
- Use **Arrow** keys to navigate to **Advanced > IT8728F Super IO Configuration**. Then, press the **Enter** key.
- Navigate to **Restore AC Power Loss** and press the **Enter** key. Select **Power On**, and press the **Enter** key to confirm.
- Press the **ESC** key to return to the Advanced menu. Navigate to **Save & Exit**, select **Save Changes and Reset**. Press the **Enter** key to confirm and restart the controller.

Your controller should now be configured to boot when the machine is powered on.

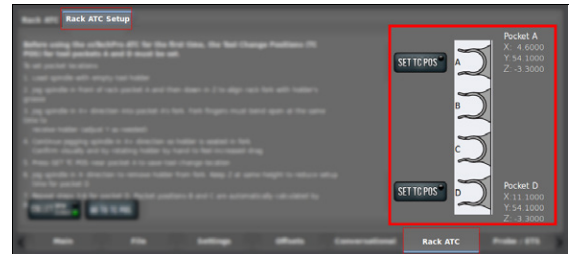
#### 3.5.5 Align the ATC Rack

Before using the automatic tool changer (ATC) for the first time, you must set the tool change positions for the rear and front tool pockets to align the ATC rack to the machine's reference position.

- Install a tool holder into the spindle.
- Jog the machine in front of the rear pocket of the ATC rack.
- Jog the spindle down to visually align the ATC fork with the tool holder's groove.
- Jog the machine in the Y+ direction to move the tool holder into the ATC rack while watching the tool fork. You want to see both forks start to bend out at the same time when the tool starts engaging the fork. Make any adjustments needed in the X direction.
- Continue jogging the machine in the Y+ direction until the holder is seated in the ATC fork. You can determine

this visually or by rotating the tool by hand. When the tool holder touches the back of the fork, you'll feel the drag increase.

- From the PathPilot interface, select **Set TC POS** below the active pocket to teach PathPilot the location for this pocket.



**Figure 3-25: Set TC POS button on Rack ATC Setup tab.**

- Jog the machine in the Y- direction to remove the tool from the ATC rack.



**Note:** Keep the Z height at the same position to help reduce the setup time for the next pocket.

- Jog the machine over to the front pocket, and repeat Steps 3-6.
- Jog the machine in the Y- direction to remove the tool from the ATC rack.
- You can make fine adjustments by directly editing the TC positions on the **Rack ATC Setup** tab. It's important to note that what's displayed are the G53 or machine coordinates. To have your machine's DRO match these values, you must:
  - Have a work coordinate active with all values set to 0, and
  - Have T0 active **or** a tool with a 0 in. offset.

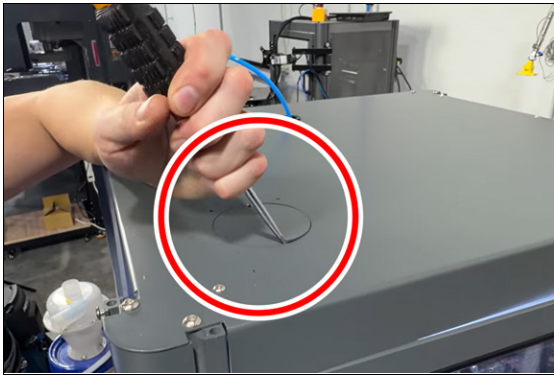
### 3.6 INSTALL ACCESSORY COMPONENTS

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

|                                                     |    |
|-----------------------------------------------------|----|
| 3.6.1 Install the Dust Collection Kit .....         | 41 |
| 3.6.2 Install the Probe .....                       | 43 |
| 3.6.3 Set up the Electronic Tool Setter (ETS) ..... | 44 |

#### 3.6.1 Install the Dust Collection Kit

1. Identify the knockout panel on the right side of the enclosure's top panel. Remove it with a flat-blade screwdriver and a dead-blow hammer.



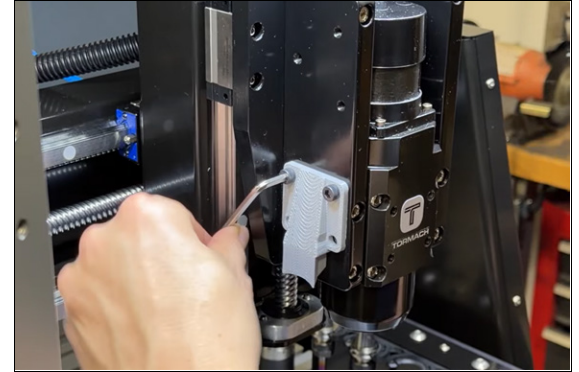
**Figure 3-26: Removing the knockout panel.**

2. Identify the hose guide. Attach it in place of the knockout panel with the four provided pan head screws. Note the direction of this part — make sure that it's pointed toward the back of the machine.



**Figure 3-27: Hose guide positioned on the top of the enclosure.**

3. Identify the dust boot and its bracket. Open the enclosure door, and attach the bracket to the left side of the spindle with the four provided M5 screws.



**Figure 3-28: Installing the dust boot bracket.**

4. Press the dust boot into the bracket (the fit is friction based).



**Figure 3-29: Pressing the dust boot into the bracket.**

5. Identify the two hose brackets. From the back of the machine, slide the quarter-turn nut of each the brackets into the slot on the right-hand side.



**Figure 3-30: Sliding the hose bracket into the slot on the back of the machine.**

6. Position one bracket toward the bottom of the machine and the other bracket toward the top. Tighten the M5 screw on each once they're in the correct position.

## 3: INSTALLATION

### 3.6 Install Accessory Components

7. Identify the hose. Feed one end through the hose guide on the top of the machine until it reaches the dust boot on the spindle. Then, clamp the other end of the hose into the two brackets that you installed on the back of the machine.
8. Identify the quick-connect piece. Thread it on to the loose end of the hose near the spindle (it's a left-handed thread).



**Figure 3-31: Threading the quick-connect piece onto the hose.**

9. Press the quick-connect piece into the dust boot.



**Figure 3-32: Connecting the hose to the dust boot.**

10. Adjust the slack on the hose so that it has enough length to reach throughout the cabinet. You can test this by closing the enclosure door and jogging the spindle.
11. The loose end of the hose connects to the dust separator. If you'd prefer, you can cut any additional length from the loose end of the hose before making this connection. Once the hose is cut to your desired length, identify the fitting for the separator's inlet. Thread it onto the loose end of the hose (it's a left-handed thread).



**Figure 3-33: Threading the inlet connector onto the hose.**

12. Attach the connector to the inlet on the dust separator.



**Figure 3-34: Attaching the hose to the dust separator.**

13. Identify the bag of universal fittings, and determine which fitting works best for your shop vacuum system. Once you've selected the fitting, attach the dust separator to your shop vacuum system.



**Note:** CAD files for the 3D-printed components in this kit are available on [tormach.com](http://tormach.com) for replacement or customization.

#### (Optional) Install the AC Contactor Box for PathPilot Control

If you have the (optional) AC contactor box, you can control a vacuum, pump, or other device using PathPilot.

1. Connect the contactor box to **AC Contactor Module** outlet on the back of the machine.



**Figure 3-35: AC Contactor Module outlet.**

2. On the contactor box, identify the **Load** and **Input** labels.
3. Connect your vacuum to the **Load** outlet, and connect the **Input** cable to a wall outlet or power strip.

To turn the vacuum on and off in PathPilot, use the following M-codes in the **MDI Line DRO** field:

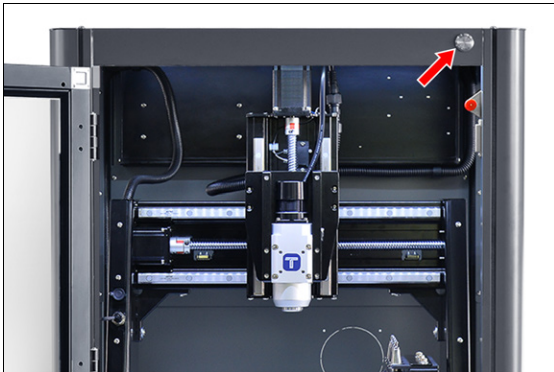
- To turn the vacuum **on**, program **M27**
- To turn the vacuum **off**, program **M28**

#### 3.6.2 Install the Probe

You'll need a 1-2-3 block (or another object of known height) and a ring gauge for this procedure. Collect them before you begin.

##### Load and Verify the Probe

1. Install the probe in a tool holder.
2. Load the probe into the spindle by pressing the **Drawbar Release** button and inserting the tool holder.



**Figure 3-36: Drawbar Release button.**

3. From the PathPilot interface, change the active tool number to 99. Tool 99 is reserved for the probe.
4. Plug the probe cable into the **Probe Control** connector on the left side of the machine's frame.



**Figure 3-37: Probe Control connector.**

5. From the PathPilot interface, on the **Probe/ETS** tab, select the **Probe Setup** tab.
6. Check probe status: lightly press the probe tip. When the probe triggers, the indicator LED in PathPilot should illuminate.

If the LED illuminates, the probe is communicating correctly with PathPilot.



**Figure 3-38: Probe Status LED on the Probe Setup tab.**

##### Set the Reference Height

1. Remove the probe from the spindle and set it aside.
2. From the PathPilot interface, set the active tool to 0.
3. Place the 1-2-3 block on the machine table with the 2-inch side facing up.
4. Jog the Z-axis down until the tool just touches the top of the gauge block.

# 3: INSTALLATION

## 3.6 Install Accessory Components

- From the **Probe/ETS** tab, on the **Probe Setup** tab, select **Set Reference Height**.



Figure 3-39: Reference height section of the probe setup.

- Leave the gauge block in place and reinstall the probe in the spindle. Then, from the PathPilot interface, change back to tool 99.
- Verify that the probe still triggers correctly when the tip is pressed.
- Align the gauge block so that the probe will contact the surface—not a hole—on its next move.
- From the PathPilot interface, on the **Probe/ETS** tab, open the **Probe Setup** tab. Then, select **Move and Set Probe Length**.

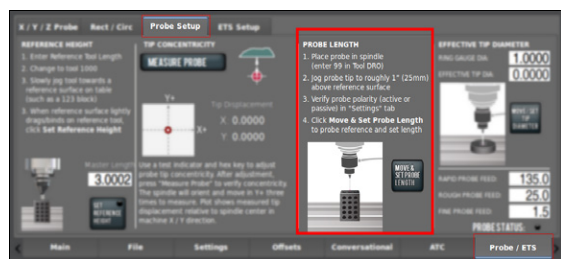


Figure 3-40: Probe length section of the probe setup.

### Set the Probe Tip Diameter

- On the ring gauge, identify the diameter that's printed on the gauge itself. For example, ours is 29.994 mm.
- Place the ring gauge on the machine table, and then jog the probe tip into the ring gauge.
- From the PathPilot interface, on the **Probe** tab, enter the diameter of the ring gauge into the **Ring Gauge Diameter** DRO field. Then, select **Move/Set Tip Diameter**.



Figure 3-41: Effective tip diameter section of the probe setup.

### 3.6.3 Set up the Electronic Tool Setter (ETS)

#### Verify Tool Setter Function

- Confirm that the machine is referenced.
- From the PathPilot interface, on the **Probe/ETS** tab, select the **ETS Setup** tab.
- Open the machine door and locate the tool setter.
- Press the tool setter pad by hand.

The **ETS Input** LED in PathPilot should illuminate, indicating that the tool setter is communicating correctly.

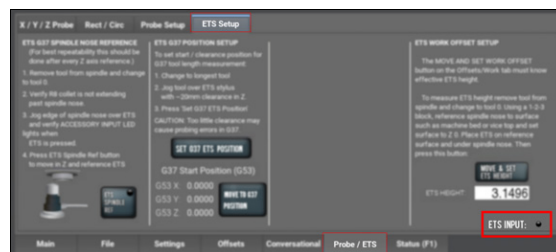


Figure 3-42: ETS Input LED on the ETS Setup tab.

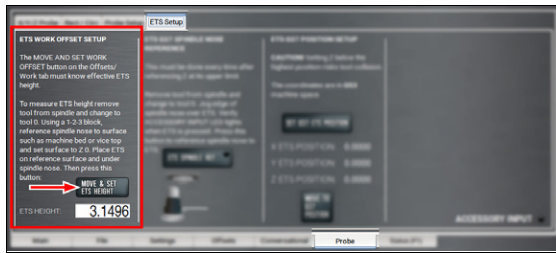
#### Set the Tool Setter Reference Height

Before you begin to use the ETS, you must first use the PathPilot interface to set its work offset.

To set the ETS height:

- Jog the spindle until it's over the ETS.
- From the PathPilot interface, set the active tool to 0.

- From the PathPilot interface, on the **Probe** tab, select the **ETS Setup** tab. Then find the **ETS Work Offset Setup** group, and select **Move & Set ETS Height**.



**Figure 3-43: Move & Set ETS Height button on the Probe tab.**

The Z-axis moves down (-Z) until the spindle nose contacts and triggers the ETS.

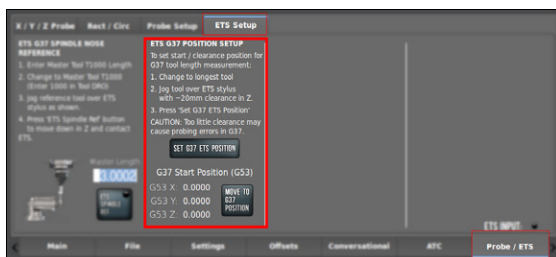
- In the **ETS Height** DRO field, verify that the length of the ETS updated.

#### Set the G37 Reference Position

- Load the longest tool that you'll be using into the spindle. From the PathPilot interface, change to that tool.

**NOTICE!** If you use a shorter tool to set the G37 position, there's a risk of tool collision when using a longer tool.

- Jog the tool so that it's about 20 mm above the tool setter's contact pad.
- From the PathPilot interface, on the **ETS Setup** tab, find the **ETS G37 Position Setup** group, and select **Set G37 ETS Position**.



**Figure 3-44: Set G37 ETS Position on the Probe tab.**

The G37 position is now set.

- Verify that the X, Y, and Z ETS position displayed in their DRO fields are accurate.



**Note:** The values displayed in these DRO fields are in G53.

## 3: INSTALLATION

### 3.7 Set Up the PathPilot Controller

#### 3.7 SET UP THE PATHPILOT CONTROLLER

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

##### 3.7.1 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**.

##### 3.7.2 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**.

##### 3.7.3 Update PathPilot

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

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

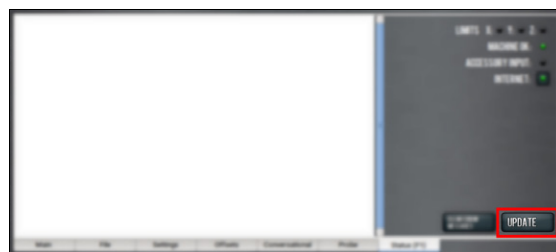


Figure 3-45: Update button on the Status tab.

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

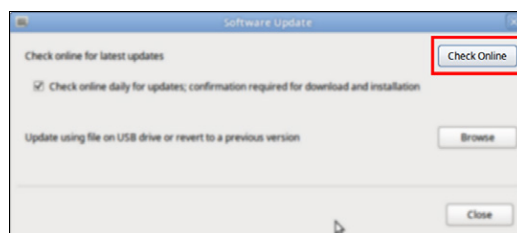


Figure 3-46: Software Update dialog box.

4. Select **Install**.



Figure 3-47: 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.

# PATHPILOT INTERFACE OVERVIEW

## IN THIS SECTION, YOU'LL LEARN:

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

---

### CONTENTS

|                                     |    |
|-------------------------------------|----|
| 4.1 About PathPilot .....           | 48 |
| 4.2 Notebook Section .....          | 49 |
| 4.3 Persistent Controls .....       | 52 |
| 4.4 Keyboard Shortcuts .....        | 54 |
| 4.5 Manage PathPilot Versions ..... | 55 |

# 4: PATHPILOT INTERFACE OVERVIEW

## 4.1 About PathPilot

### 4.1 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.

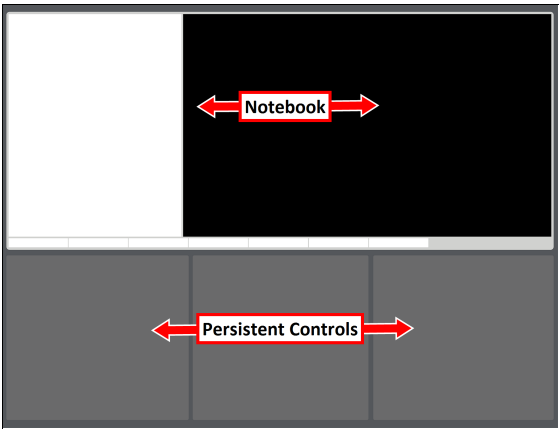
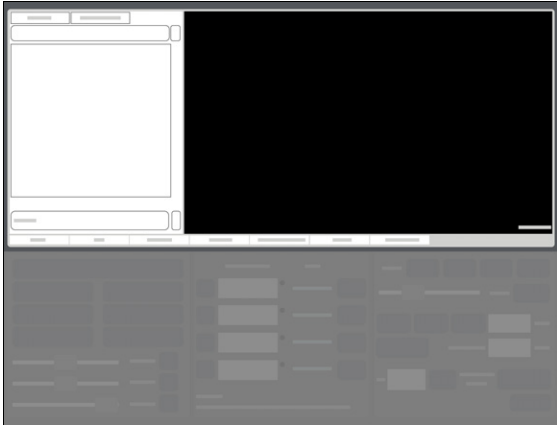


Figure 4-1: Sections in the PathPilot interface.

### 4.2 NOTEBOOK SECTION

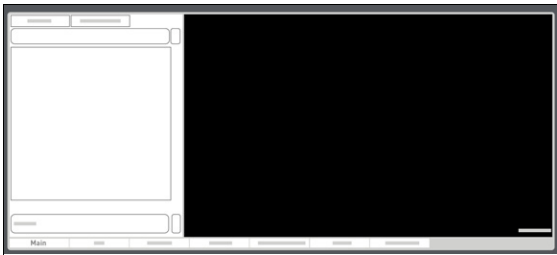


**Figure 4-2: Notebook section.**

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

|                                |    |
|--------------------------------|----|
| 4.2.1 Main Tab .....           | 49 |
| 4.2.2 File Tab .....           | 49 |
| 4.2.3 Settings Tab .....       | 49 |
| 4.2.4 Offsets Tab .....        | 50 |
| 4.2.5 Conversational Tab ..... | 50 |
| 4.2.6 Probe/ETS Tab .....      | 50 |
| 4.2.7 Status Tab .....         | 50 |

#### 4.2.1 Main Tab



**Figure 4-3: 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 59); "Close the Current Program" (page 59).
- Send G-code commands directly to the machine using the Manual Data Input (MDI) Line DRO field.  
For information, see "Manually Enter Commands" (page 97).

- 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 61); "Expand the G-Code Tab" (page 60); "Change the View of the Tool Path Display" (page 62).
- Make and restore backup files of your settings.  
For information, see "Create Backup Files" (page 103); "Restore Backup Files" (page 104).

#### 4.2.2 File Tab

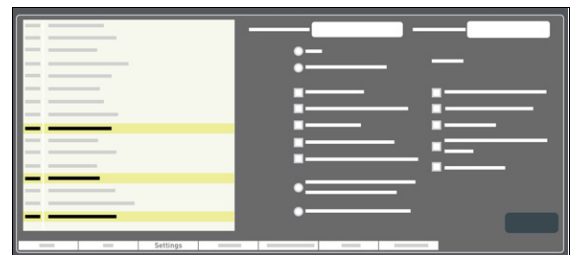


**Figure 4-4: File tab.**

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 58).
- Edit G-code files.  
For information, see "Edit G-Code" (page 59).
- Load .nc files into PathPilot to run a program.  
For information, see "Load G-Code" (page 114).
- Move files within the system.  
For information, see "Preview G-Code Files" (page 58); "Manage System Files" (page 103).

#### 4.2.3 Settings Tab



**Figure 4-5: 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 74).

## 4: PATHPILOT INTERFACE OVERVIEW

### 4.2 Notebook Section

- Change the screen's layout orientation (landscape or portrait).  
For information, see "Change the Screen Orientation" (page 75).
- Configure PathPilot for the accessories you're using.  
For information, see "Enable the On-Screen Keyboard" (page 77); "Enable the USB M-Code I/O Interface Kit" (page 78); "Use a USB Camera" (page 78).
- Turn on feeds and speeds suggestions when using conversational programming.  
For information, see "Enable Feeds and Speeds Suggestions in Conversational Routines" (page 78).
- Specify the way in which you want to use a G30 move.  
For information, see "Limit G30 Moves" (page 77).
- Identify the available G-code modes that you can use.  
For information, see "View Available G-Code Modes" (page 88).

#### 4.2.4 Offsets Tab



**Figure 4-6: Offsets tab.**

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

- Import and export .csv files of your tool table.  
For information, see "Import and Export the Tool Table" (page 105).
- Work with a table of tool descriptions and tool offsets.  
For information, see "Set Tool Length Offsets" (page 115).
- Use an Electronic Tool Setter (ETS) to measure tools.  
For information, see Use an to Measure Tools.
- Preset a G30 position.  
For information, see "Use a G30 Position" (page 96).
- Read the currently programmed work offsets.  
For information, see "View Work Offsets" (page 88).

#### 4.2.5 Conversational Tab



**Figure 4-7: Conversational tab.**

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

- Use different functions to create simple G-code programs in PathPilot.  
For information, see "About Conversational Programming" (page 63).

#### 4.2.6 Probe/ETS Tab

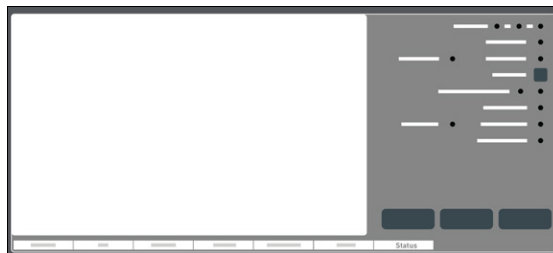


**Figure 4-8: Probe tab.**

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

- Configure and control a probe to help perform certain functions.  
For information, see "Use a Probe with PathPilot" (page 82).
- Configure and control an Electronic Tool Setter (ETS) to help perform certain functions.  
For information, see Use an to Measure Tools.

#### 4.2.7 Status Tab



**Figure 4-9: Status tab.**

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

- View diagnostic machine information.
- Read error messages.
- Configure your internet connection.  
For information, see "Enable an Internet Connection" (page 74).
- Update or install a previous version of PathPilot.  
For information, see "Manage PathPilot Versions" (page 55).

# 4: PATHPILOT INTERFACE OVERVIEW

## 4.3 Persistent Controls

### 4.3 PERSISTENT CONTROLS



Figure 4-10: Persistent Controls section.

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

|                                 |    |
|---------------------------------|----|
| 4.3.1 Program Control Area..... | 52 |
| 4.3.2 Position Status Area..... | 52 |
| 4.3.3 Manual Control Area.....  | 52 |

#### 4.3.1 Program Control Area



Figure 4-11: Program Control area.

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

- Reset the machine.  
For information, see "Bring the Machine Out of Reset" (page 109).
- Start, stop, or pause a G-code program.  
For information, see "Start a Program" (page 91); "Stop Machine Motion" (page 91); "Use the Feed Hold Function" (page 93).
- Turn coolant on or off.  
For information, see "Operate the Coolant Pump" (page 120).
- Use overrides to change the feed rate, spindle speed, and maximum velocity.

For information, see "Use the Feed Rate Override Function" (page 93); "Use the Maxvel Override Function" (page 94); "Use the Spindle Override Function" (page 95).

- Manually control a G-code program.  
For information, see "Use M01 Break Mode" (page 94); "Use Single Block Mode" (page 95).

#### 4.3.2 Position Status Area



Figure 4-12: Position Status area.

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

- Reference the machine axes.  
For information, see "Reference the Machine" (page 110).
- Create work offsets.  
For information, see "Set Work Offsets" (page 118).
- Understand how you're jogging the machine.  
For information, see "View the Active Axis to Jog" (page 89); "View the Current Machine Position" (page 90); "View the Distance to Go" (page 92).
- Quickly determine which G-code modes are active.  
For information, see "View the Active G-Code Modes" (page 92).

#### 4.3.3 Manual Control Area



Figure 4-13: Manual Control area.

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

- Move the machine axes.  
For information, see "Jog the Machine" (page 111).

- Change the spindle speed or feed rate.  
For information, see "Change the Spindle Speed" (page 96); "Change the Feed Rate" (page 96).
- View or edit information about the current tool.  
For information, see "Change the Tool Number" (page 96); "Use a G30 Position" (page 96) "View the Tool Length" (page 97).

# 4: PATHPILOT INTERFACE OVERVIEW

## 4.4 Keyboard Shortcuts

### 4.4 KEYBOARD SHORTCUTS

The following table lists the keyboard shortcuts in PathPilot.

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

### 4.5 MANAGE PATHPILOT VERSIONS



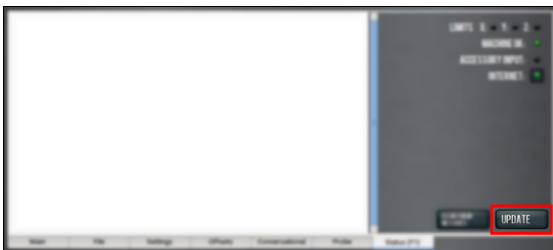
**Note:** To update PathPilot on a CE machine, you must first confirm that the machine is in setup mode.

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

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

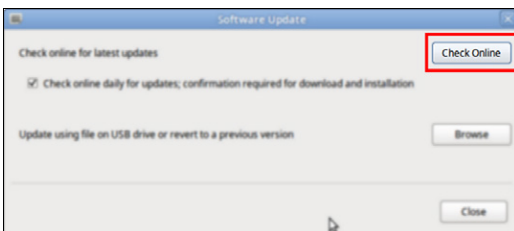
#### 4.5.1 Download and Install an Update File from the Controller

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 4-14: Update button on the Status tab.**

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



**Figure 4-15: Software Update dialog box.**

4. Select **Install**.



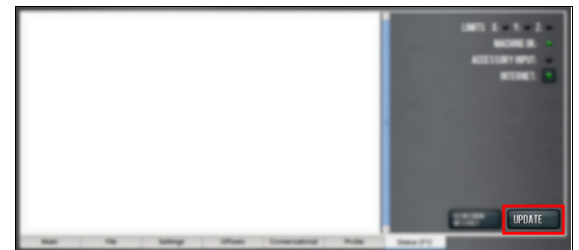
**Figure 4-16: Install button on the Software Update dialog box.**

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

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

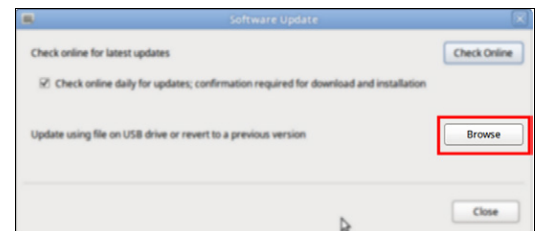
#### 4.5.2 Install an Update File from a USB Drive

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 4-17: Update button on the Status tab.**

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



**Figure 4-18: Software Update dialog box.**

# 4: PATHPILOT INTERFACE OVERVIEW

## 4.5 Manage PathPilot Versions

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



Figure 4-19: Browse dialog box.

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

### 4.5.3 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**.

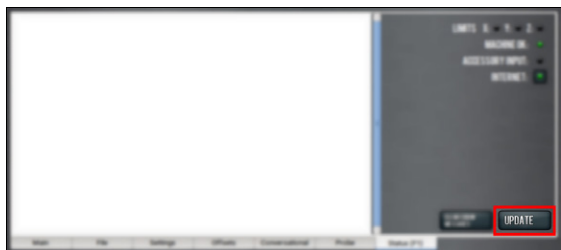


Figure 4-20: Update button on the Status tab.

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

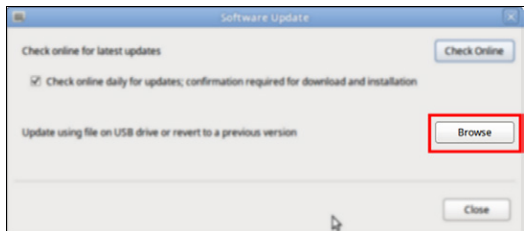


Figure 4-21: Software Update dialog box.

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

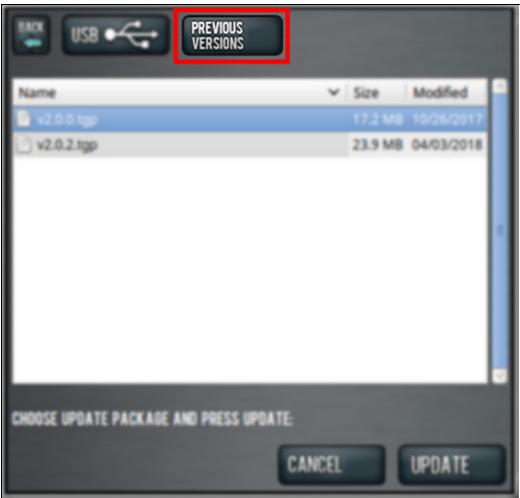


Figure 4-22: Browse dialog box.

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

# PATHPILOT TOOLS AND FEATURES

## IN THIS SECTION, YOU'LL LEARN:

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

---

### CONTENTS

|                                            |     |
|--------------------------------------------|-----|
| 5.1 Create and Load G-Code Files .....     | 58  |
| 5.2 Machine Settings and Accessories ..... | 74  |
| 5.3 Set Up G-Code Programs .....           | 82  |
| 5.4 Run G-Code Programs .....              | 89  |
| 5.5 Control G-Code Programs .....          | 93  |
| 5.6 System File Management .....           | 103 |

5: PATHPILOT TOOLS AND FEATURES

5.1 Create and Load G-Code Files

5.1 CREATE AND LOAD G-CODE FILES

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

|                                      |    |
|--------------------------------------|----|
| 5.1.1 Load G-Code                    | 58 |
| 5.1.2 Edit G-Code                    | 59 |
| 5.1.3 Read G-Code                    | 60 |
| 5.1.4 Use Conversational Programming | 63 |

5.1.1 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**.

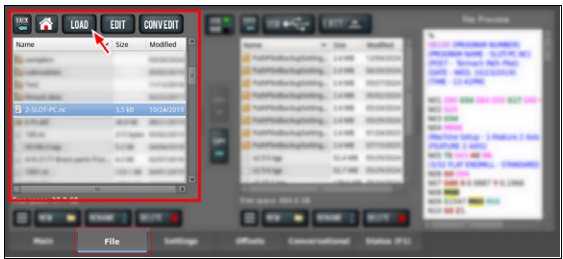


Figure 5-1: Controller Files window on the File tab.

**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](http://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).

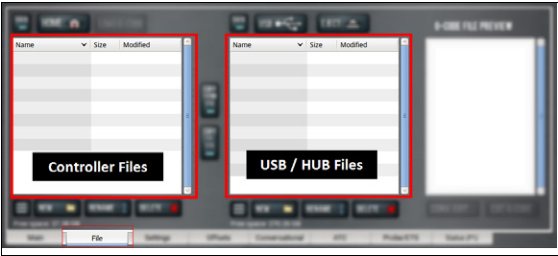


Figure 5-2: File tab.

**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 5-3: File tab.

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

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

Preview G-Code Files

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

To preview G-code files:

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

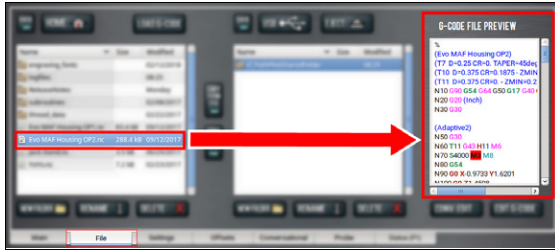


Figure 5-4: File tab.

### 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.

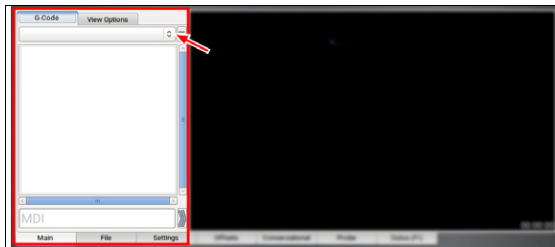


Figure 5-5: Recent Files menu on the Main tab.

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**.

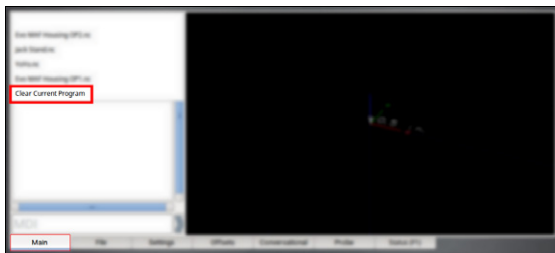


Figure 5-6: Recent Files menu on the Main tab.

The currently loaded G-code program closes.

### 5.1.2 Edit G-Code

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

- **Edit G-Code with a Text Editor**..... 59
- **Edit G-Code with Conversational Programming**..... 59

### 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**.

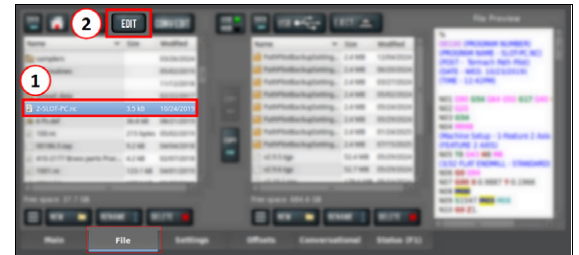


Figure 5-7: Edit G-code button on the File tab.

The file opens in a text editor.

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

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

### Edit G-Code with Conversational Programming

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

To edit G-code with conversational programming:

5: PATHPILOT TOOLS AND FEATURES

5.1 Create and Load G-Code Files

- 1. From the **File** tab, select the .nc file.
  - 2. Select **Conv. Edit**.
- The file opens in a job assignment editor window: the program's job assignments are on the left and a preview of the program is on the right.

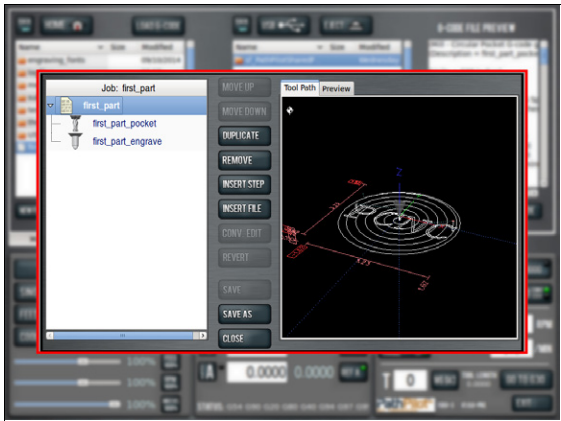


Figure 5-8: Job assignment editor.

- 3. Edit the file contents as needed. Do any of the following:
- |                                   |    |
|-----------------------------------|----|
| Change the Step Order.....        | 60 |
| Create a New Job Assignment.....  | 60 |
| Load an Existing G-Code File..... | 60 |
| Edit a Job Assignment.....        | 60 |
- 4. Select **Save**.
- The G-code program file is updated.

Change the Step Order

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

Create a New Job Assignment

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

Load an Existing G-Code File

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

Edit a Job Assignment

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

Tips

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

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

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

5.1.3 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.....                    | 60 |
| Search in the Code.....                       | 61 |
| Set a New Start Line.....                     | 61 |
| Change the View of the Tool Path Display..... | 62 |

Expand the G-Code Tab

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

To expand the G-Code tab:

- Select the **Window Expander**.

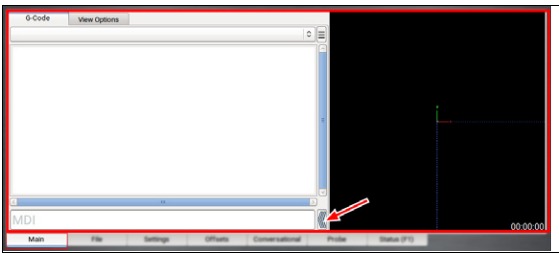


Figure 5-9: Window Expander on the Main tab.

The **Tool Path** display shrinks.

About the G-Code Tab

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

can make the G-Code tab larger. For information, see "Expand the G-Code Tab" (on the previous page).

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

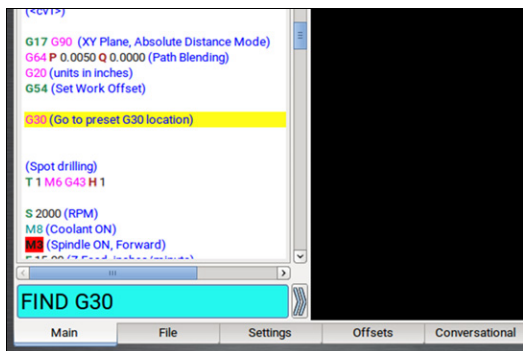
- **Green Line** Indicates the start line (the line from which PathPilot starts the program).  
To change the start line, go to "Set a New Start Line" (below).
- **Orange Line** Indicates the line of code that PathPilot is currently executing.

### Search in the Code

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

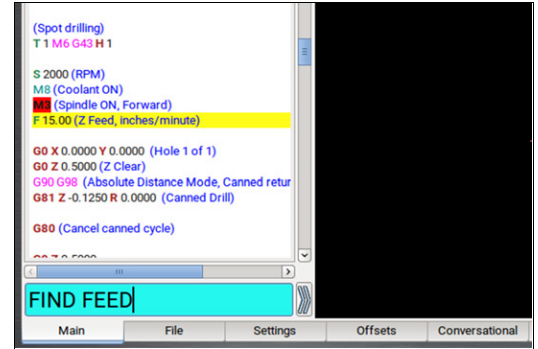
To search in the code:

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



**Figure 5-10: Search for a text command.**

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



**Figure 5-11: Search for a feed command.**

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



**Note:** The find command is not case-sensitive.

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



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

### Set a New Start Line

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

1. From the **Main** tab, on the **G-Code** tab, do one of the following:

5: PATHPILOT TOOLS AND FEATURES

5.1 Create and Load G-Code Files

- Right-click any line in the program.

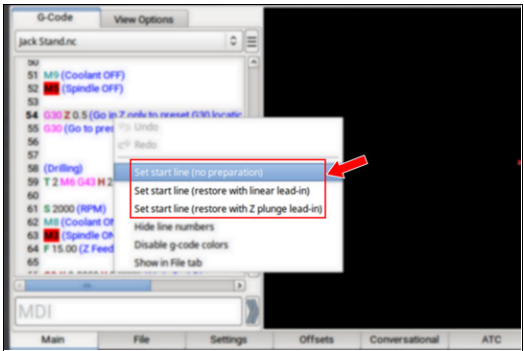


Figure 5-12: Accessing the Options menu by right-clicking.

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

Lead-In Moves

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



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



EXAMPLE

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

- Set start line (restore with linear lead-in)** Perform a tool change (as required). The machine rapids in X 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 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.

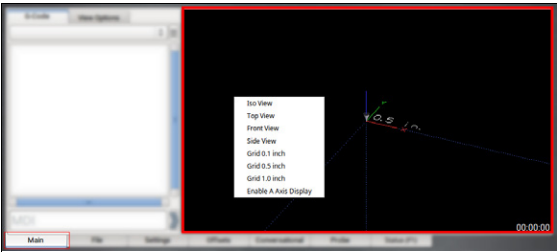


Figure 5-13: Tool Path display on the Main tab.

- Select the **View Options** tab.

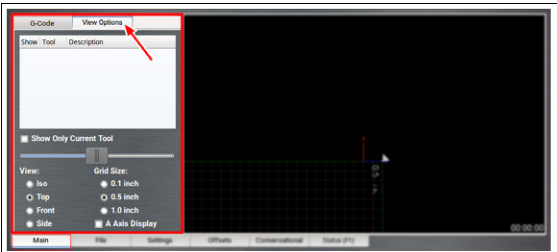


Figure 5-14: View Options tab on the Main tab.

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.

There are four available views:

- Front
- Iso
- Side
- Top

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

- **G20 Mode** 1/2 in. intervals
- **G21 Mode** 10 mm intervals

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

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



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

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

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

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

### 5.1.4 Use Conversational Programming

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

|                                              |    |
|----------------------------------------------|----|
| About Conversational Programming.....        | 63 |
| Create a Face on a Part .....                | 63 |
| Create a Profile on a Part .....             | 65 |
| Create a Pocket on a Part .....              | 66 |
| Create Hole Locations on a Part (Drill)..... | 68 |

|                                        |    |
|----------------------------------------|----|
| Create Threads on a Part .....         | 70 |
| Create Text to Engrave on a Part ..... | 71 |
| Import a DXF File .....                | 72 |

### About Conversational Programming

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

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



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

The Conversational tab is divided into two sections:

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



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

- Parameters specific to each operation, like part geometry.

### Create a Face on a Part

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

#### Before You Begin

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

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

# 5: PATHPILOT TOOLS AND FEATURES

## 5.1 Create and Load G-Code Files

To create a face on a part:

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

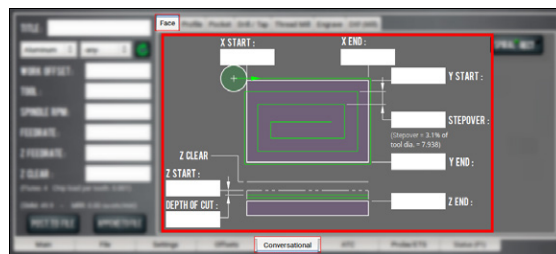


**Figure 5-15: Conversational DROs on the Face tab.**

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



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



**Figure 5-16: Program-specific DROs on the Face tab.**

### About Facing

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

### Facing in PathPilot

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

During a facing routine, PathPilot does the following:

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



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

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

For information on using conversational programming in PathPilot to face a part, see "Create a Face on a Part" (page 63).

### Facing Reference

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

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

### Create a Profile on a Part

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

### Before You Begin

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

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

To create a profile on a part:

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

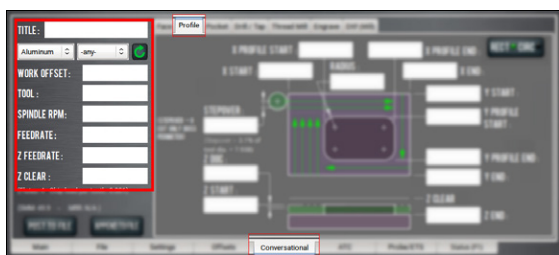


Figure 5-17: Conversational DROs on the Profile tab.

3. Work through the program-specific DRO fields:
  - a. In the **X Start** DRO field and the **X End** DRO field, type the location of the workpiece edges.
  - b. In the **Y Start** DRO field and the **Y End** DRO field, type the location of the workpiece edges.

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



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

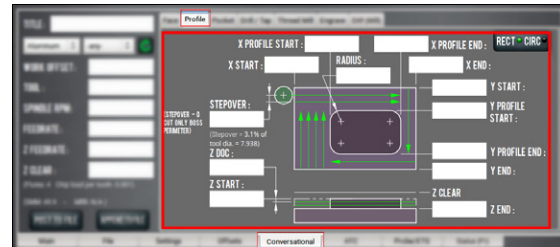


Figure 5-18: Program-specific DROs on the Profile tab.

### About Profiling

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

### Profiling in PathPilot

When using a profile routine, each tool pass along the X-/Y-axis begins off to the side of the workpiece to avoid plunging into the workpiece. To compensate for this procedure, PathPilot sets lead-in tool paths outside of the workpiece using the part's work offsets, the tool's diameter, and the predetermined stepover value. PathPilot also adjusts the depth

# 5: PATHPILOT TOOLS AND FEATURES

## 5.1 Create and Load G-Code Files

of cut to make sure each tool pass has the same depth, rather than cutting a short depth on the last pass of the program. When you're creating a profile on a part using conversational programming, PathPilot does the following in the order listed:

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



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

2. Makes a rapid movement to the beginning of the section.
3. If required, makes a tool path around the perimeter of the boss to cut the programmed radii.



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

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



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

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

### Profiling Reference

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

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

### Create a Pocket on a Part

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

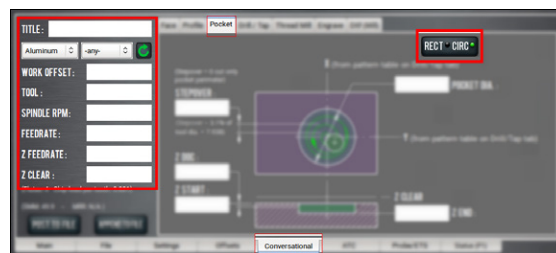
### Before You Begin

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

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

To create a circular pocket on a part:

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



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



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

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



# 5: PATHPILOT TOOLS AND FEATURES

## 5.1 Create and Load G-Code Files

listed:

1. Uses a helical entry into the pocket center.
2. Makes a hole that is two times the diameter of the tool diameter in the pocket center.
3. Makes a spiral cut out from the pocket center to the pocket diameter.
4. Makes a cut around the perimeter of the pocket.

### About Rectangular Pockets

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

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

### Create Hole Locations on a Part (Drill)

Using conversational programming, you can program PathPilot to make multiple holes on a part. For information, see About Drilling.

#### Before You Begin

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

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

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

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

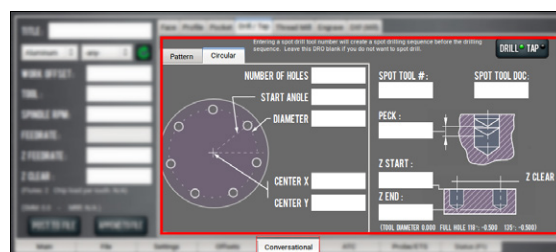


Figure 5-19: Drill tab on the Conversational tab.

2. From the **Drill** tab, select the **Circular** tab.
3. In the **Number of Holes** DRO field, type the number of holes (greater than 0) required for the pattern.
4. In the **Start Angle** DRO field, type the value of the angle from angle 0 (from **-90** to **90**).
5. In the **Diameter** DRO field, type the size of the circular pattern (as defined by a line through the center point of each hole).
6. In the **Center X** DRO field and the **Center Y** DRO field, type the location for the center of the hole.
7. Go to "Create a Drilling Sequence" (on the next page).

Use the Location table to make a list of X and Y locations for each hole using the same tool, the same Z location, and the same values in the Conversational DROs group.

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

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

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

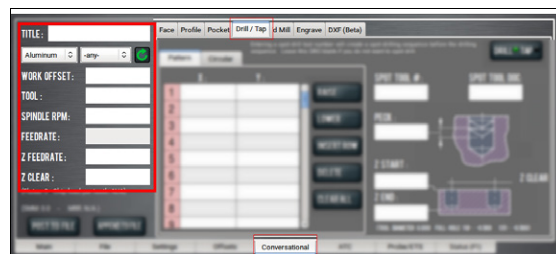


Figure 5-20: Drill tab on the Conversational tab.

2. From the **Drill** tab, select the **Pattern** tab.

3. In the **Location** table, select the row, then select the cell to edit.  
Holes indicated in the **Location** table are completed from top to bottom.
4. Go to "Create a Drilling Sequence" (below).

### Tips

- To rearrange the row order, select a row, and then select **Raise** or **Lower**.
- To clear all values in the table, select **Clear All**.
- To make sure that the value is a valid number, leave the cell. If the value isn't a valid number, it's erased, and an error displays on the **Status** tab.
- Before posting the file, make sure that there are none of the following:
  - X values without a Y value
  - Y values without an X value
  - An empty row before the last row with values

### About Drilling

PathPilot has the following options to define the locations:

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

For information on using conversational programming in PathPilot to drill holes, see .

During a drilling routine, PathPilot uses one of the following canned G-code cycles, depending on the entries in each DRO field:

- **G81** Drill
- **G82** Drill with dwell
- **G83** Drill with peck



**Note:** The features can't be combined, because peck cancels dwell.

### Create a Drilling Sequence

The Drill tab uses one of the following canned G8x cycles to drill a hole at each identified location:

- **G81** Drill
- **G82** Drill with dwell
- **G83** Drill with peck



**Note:** The features can't be combined, because peck cancels dwell.

Hole depth is usually defined as the full diameter portion of the hole, so you may need to consider the Z length from the drill point to the corner.

1. In the **Spot Tool #** DRO field, type the number of the spot drill.

If a valid tool number is used in the **Spot Tool #** DRO field, PathPilot makes a spot drilling sequence before the drilling sequence. In the spot drilling sequence, PathPilot uses the values indicated in the **Feedrate** DRO field, the **Spindle RPM** DRO field, and the **Z Clear** DRO field.



**Note:** To skip a spot drilling sequence, leave the **Spot Tool #** DRO field blank.

2. In the **Spot Tool DOC** DRO field, type the depth of cut for the spot drilling sequence.



**Note:** To skip a spot drilling sequence, leave the **Spot Tool DOC** DRO field blank.

3. In the **Peck** DRO field, type the distance for each peck.



**Note:** Any entry greater than **0** typed in the **Peck** DRO field replaces G81 with G83 in the G-code. For information, see "Drilling Reference" (below).

4. Set the parameters for the hole in the Z range:
  - a. In the **Z Start** DRO field, type the location for the first Z pass.
  - b. In the **Z End** DRO field, type the location for the last Z pass.

### Drilling Reference

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

- **Start Angle** Specifies the angle from angle 0. Angle 0 is a base (horizontal) line from the center point going right (east) to the circumference. The angle from the base line can be:

# 5: PATHPILOT TOOLS AND FEATURES

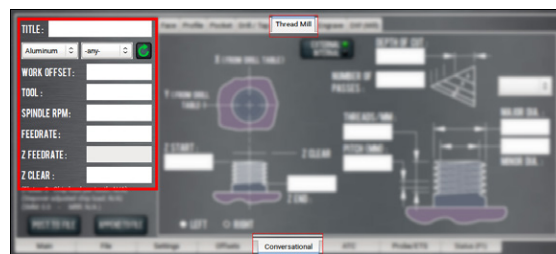
## 5.1 Create and Load G-Code Files

- Positive or negative
- Up to 90 degrees (or -90 degrees)  
A negative angle produces a clockwise rotation; a positive angle produces a counterclockwise rotation.
- Rotates the pattern either clockwise or counterclockwise  
For example, to create a hex pattern with flats on the top and bottom, use a value of 0 in the Start Angle DRO field. To create a hex pattern with flats on the left and right sides, use a value of 30 (or -30) in the Start Angle DRO field.

- **Peck** The value isn't adjusted: the G83 routine feeds at the Z Feedrate, starting from Z Clear down to the Peck distance, then rapid retracts to Z Clear, and rapid returns to start the next Peck.



**Note:** The first and last peck will likely be shorter than the value typed into the Peck DRO field.



**Figure 5-21: Conversational DROs on the Thread Mill tab.**

3. Depending on the type of threads, do one of the following:
  - For external threading (creating threads on a boss, for example), select **External**.
  - For internal threading (creating threads inside a hole, for example), select **Internal**.

### Create Threads on a Part

Using conversational programming, you can program PathPilot to take helical tool paths around a boss or inside a hole to create external or internal threads. For information, see "About Thread Milling" (on the next page).

#### Before You Begin

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

- The auto-filled DRO field values assume a full-form threaded tool. If you're using a fine-point threaded tool to cut coarse threads, you must modify the root diameter to account for the tool's smaller nose radius.
- The value used in the **Z Clear** DRO field must be such that it can clear any blockages in the path between the end of one Z pass and the beginning of the next.

To create threads on a part:

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

4. Work through the program-specific DRO fields:
  - a. From the **Quick Reference** drop-down, select the thread size.  
The **Threads/In** DRO field, **Pitch (In)** DRO field, **Major Dia.** DRO field, and **Minor Dia.** DRO field auto-fill.



**Note:** The threads listed follow the current unit setting: either inches (G20) or millimeters (G21).

- b. In the **Z Start** DRO field, type the location of the first Z pass (where you want the thread to start).
- c. In the **Z End** DRO field, type the location of the last Z pass (where you want the thread to end).

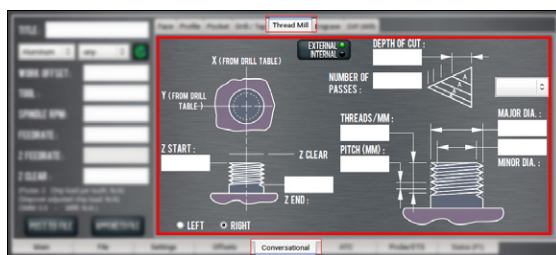


**Note:** The tool actually goes beyond the value entered in the **Z End** DRO field, due to the cutting tip width and the Z component of the compound feed angle and thread depth.

- d. In the **Depth of Cut** DRO field or the **Number of Passes** DRO field, type a value to represent the amount of material to remove in each helical pass. For information, see "Thread Milling Reference" (below).



**Note:** The **Depth of Cut** DRO field and the **Number of Passes** DRO field are linked — after you type a value in one and press **Enter** on the keyboard, the other is auto-filled.



**Figure 5-22: Program-specific DROs on the Thread Mill tab.**

### About Thread Milling

Thread milling is used to make helical tool paths on a part — either externally, like on a boss, or internally, like in a hole. The right-handed threads are based on pitch, diameter, and length.

### Thread Milling in PathPilot

For information on using conversational programming in PathPilot to do thread milling, see "Create Threads on a Part" (on the previous page).

### Thread Milling Reference

PathPilot uses the following terms when creating a thread mill on a part using conversational programming:

- **Major Dia. and Minor Dia.** Set the start and end diameter of the thread peak and valley.
- **Depth of Cut** Sets the amount of material cut in each helical pass — the value is the distance (the change in radius) the tool is fed on the first pass. This first pass cuts a triangular area which is related to the chip load. Subsequent cut depths are set to cut the same amount of area, so the linear feed gets smaller for each pass. The tool is also fed in on a compound angle of 30 degrees, keeping the cuts to one face of the tool.



**Note:** The Depth of Cut DRO field and the Number of Passes DRO field are linked — after you type a value in one and press **Enter** on the keyboard, the other is auto-filled.

- **Z Clear** The location the tool moves or retracts to when starting or ending a Z pass.

### Create Text to Engrave on a Part

Using conversational programming, you can program PathPilot to engrave a single line of text on a part. For example, you could use this feature to engrave True Type stick or outline fonts into things like simple plaques, control panels, or data plates. For information, see "About Engraving" (on the next page).

To create an engraving sequence on a part:

5: PATHPILOT TOOLS AND FEATURES

5.1 Create and Load G-Code Files

- 1. From the **Conversational** tab, select the **Engrave** tab.
- 2. From the **Conversational DROs** group, set the parameters for the engraving operation.

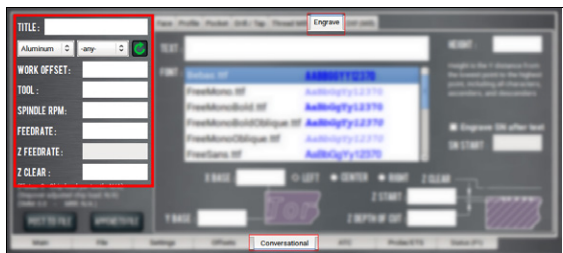


Figure 5-23: Conversational DROs on the Engrave tab.

- 3. Work through the program-specific DRO fields:
  - a. In the **Text** DRO field, type the desired text to engrave.  
The text appears in the **Preview** window.
  - b. From the **Font** list, select the desired font for the engraved text.

**Note:** To search the **Font** list, select a font and begin typing the font's name. PathPilot scrolls to the result.

The **Preview** window updates to display the font selected.

**Note:** You can add True Type font files to the PathPilot controller. For information, see "Transfer Files to and From the Controller" (page 58).

- c. Use the **Alignment** radio buttons to select the desired alignment of the text: either **Left**, **Center**, or **Right**.  
The **Preview** window updates to display the font selected.
- d. In the **Height** DRO field, type the distance in the Y direction from the top of the text to the bottom of the text.  
PathPilot uses the height value, along with font data, to calculate scale for character paths in the G-code program.

**Note:** The value typed in the **Height** DRO field includes ascenders and descenders, but no the tool cutting diameter. For a more accurate value, subtract the diameter from the overall height.

- e. In the **X Base** DRO field and the **Y Base** DRO field, type the location of the left side of the first character's baseline.

**Note:** If any characters in the text have descenders, like **y** or **g**, they extend below the baseline.

- f. In the **Z Start** DRO field, type the location of the surface on which to engrave.
- g. In the **Z Depth of Cut** DRO field, type the desired depth the that the cutting tool is fed into the workpiece.
- h. In the **Z Clear** DRO field, type the value for the tool to move or retract to — in the **Z** direction — at the start and end of the engraving sequence and between characters.

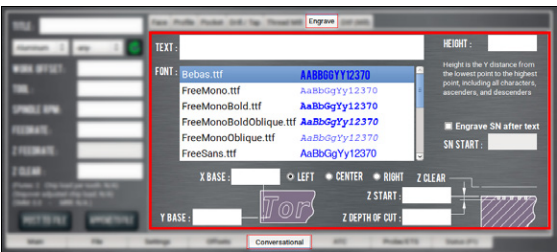


Figure 5-24: Program DROs on the Engrave tab.

About Engraving

Engraving is used to engrave a single line of text cut in a single horizontal pass along the X-axis of a part.

Import a DXF File

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

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

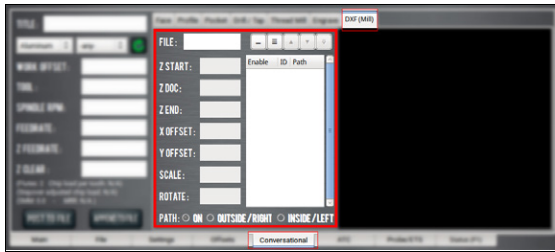


Figure 5-25: DXF tab on the Conversational tab.

2. Select the **File** DRO field.  
The **File Selector** dialog box opens.
3. Select the .dxf file, and then select **Open**.
4. The shapes from the selected file are loaded into the **Preview** window.



**Note:** The .dxf file must already be transferred to the PathPilot controller. For information, see "Transfer Files to and From the Controller" (page 58).

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

5: PATHPILOT TOOLS AND FEATURES
5.2 Machine Settings and Accessories

5.2 MACHINE SETTINGS AND ACCESSORIES

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

5.2.1 Enable an Internet Connection ..... 74
5.2.2 Change the Network Name ..... 74
5.2.3 Change the Screen Orientation ..... 75
5.2.4 Disable Limit Switches ..... 76
5.2.5 Limit G30 Moves ..... 77
5.2.6 Enable the On-Screen Keyboard ..... 77
5.2.7 Enable the USB M-Code I/O Interface Kit ..... 78
5.2.8 Enable Tooltips ..... 78
5.2.9 Enable Feeds and Speeds Suggestions in Conversational Routines ..... 78
5.2.10 Use a USB Camera ..... 78

5.2.1 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 5-26: Internet button on the Settings tab.

The Network Configuration dialog box displays.

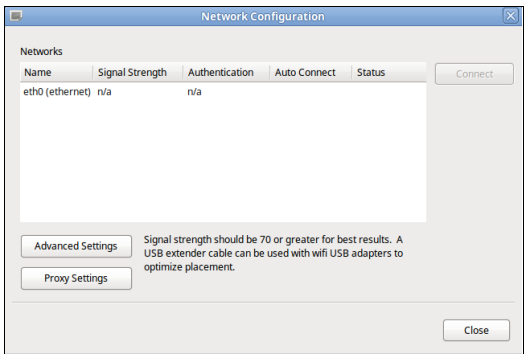


Figure 5-27: Network Configuration dialog box.

- 2. From the Network Configuration dialog box, in the Networks list, select the network you want to use. Then,

select Connect.

Note: Wi-Fi connection signal strengths are indicated on a scale of 0 to 100, with 100 being the strongest. PathPilot continually refreshes the signal levels to help you find the best placement for your Wi-Fi network adapter. Ethernet connections are indicated by a prefix in the following format: eth[NUMBER]. For example, eth1.

The PathPilot operating system connects to the internet using the network you specified. It continues to detect and connect to the Wi-Fi network, even after power cycles.

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.

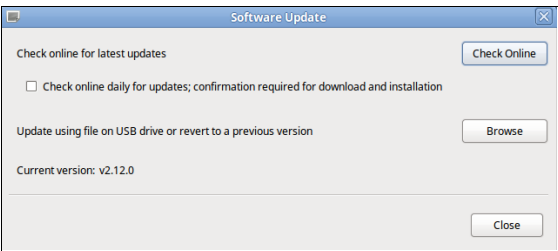


Figure 5-28: Software Update dialog box.

- 2. From the Software Update dialog box, select the Check online daily for updates; confirmation required for download and installation checkbox.
- 3. Select Close.

When future updates are available, the Status tab displays a notification.

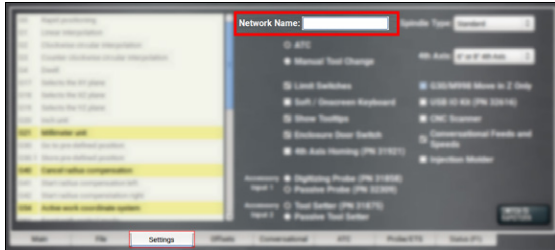
5.2.2 Change the Network Name

If you're connected to a network using either the Ethernet jack or the (optional) Wireless Network Adapter (PN 38207), the PathPilot controller appears on your network as network-

**attached storage.** The default network name of the controller is **TORMACHPCNC**.

To change the network name:

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



**Figure 5-29: Network Name field on the Settings tab.**



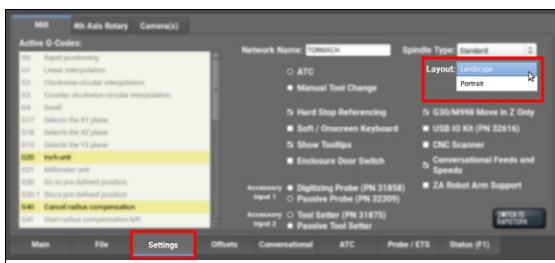
**Note:** The network name must be unique within your network.

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

### 5.2.3 Change the Screen Orientation

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

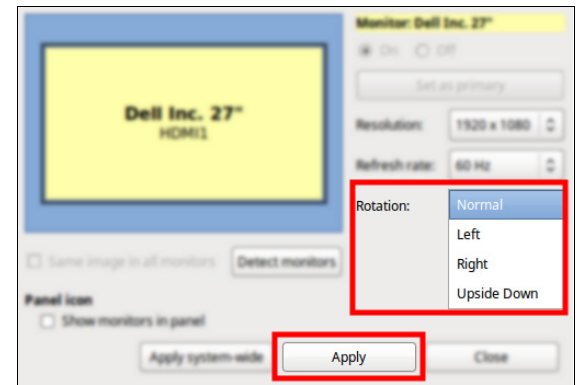
1. From the PathPilot interface, on the **Settings** tab, select **Portrait** from the **Layout** drop-down menu. Restart the controller.



**Figure 5-30: Layout drop-down menu on the Settings tab.**

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

again.



**Figure 5-31: 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.

5: PATHPILOT TOOLS AND FEATURES

5.2 Machine Settings and Accessories

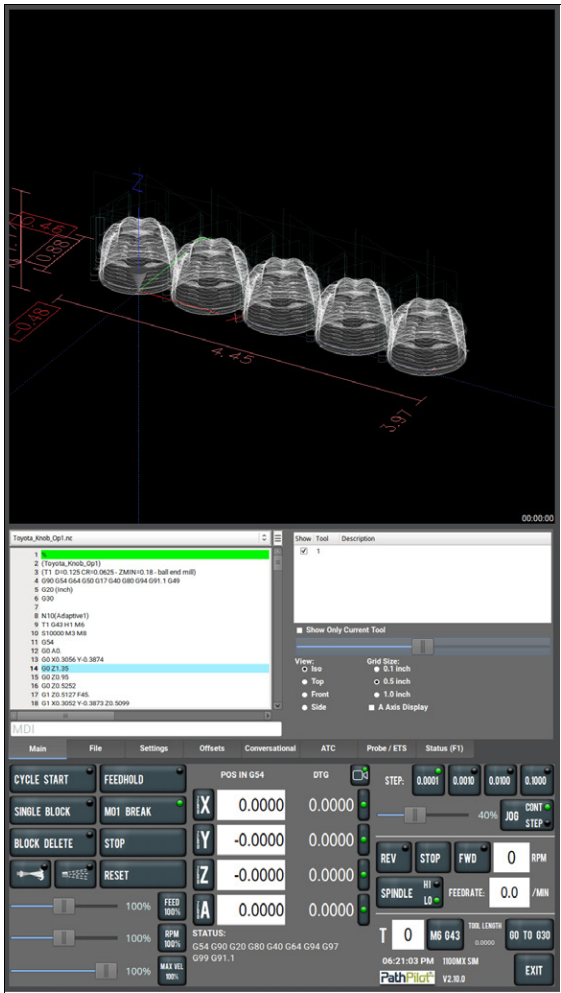


Figure 5-32: Tool Path window in portrait screen layout.

- 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.

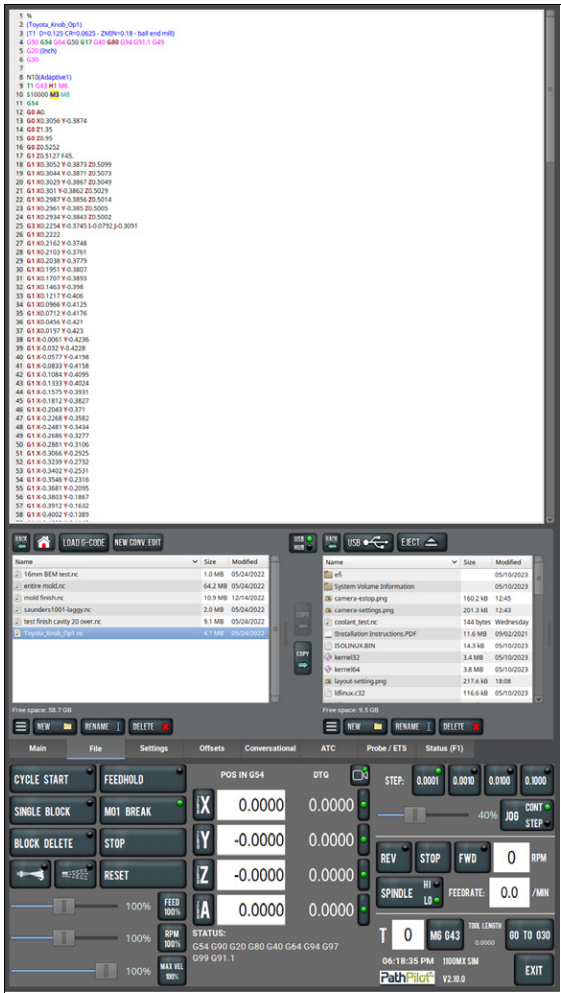


Figure 5-33: File tab G-code preview in portrait screen layout.

5.2.4 Disable Limit Switches

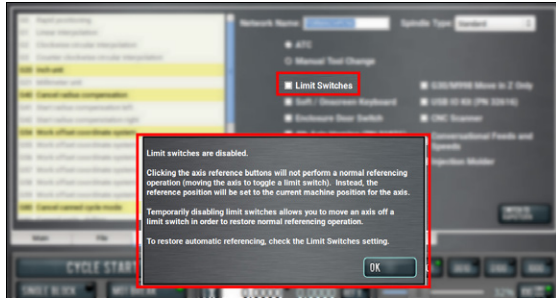
To provide a temporary workaround for a malfunctioning limit switch circuit, you can disable the limit switches. For information, see "About Limit Switches" (on the next page).



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

To disable limit switches:

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



**Figure 5-34: Limit Switches checkbox on the Status tab.**

2. Select **OK**.

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



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

### About Limit Switches

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

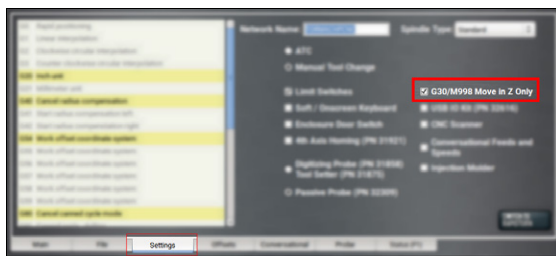
If the checkbox is cleared, the machine completes a unique referencing procedure after selecting Ref X, Ref Y, Ref Z, and Ref A: rather than moving each axis to the end of its travel, the reference position is set as the machine's current position. This is useful for troubleshooting: if the limit switches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

### 5.2.5 Limit G30 Moves

You can limit G30 moves so that only the Z-axis moves. For information, see "About G30" (page 97).

To limit G30 moves:

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



**Figure 5-35: 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 96).

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).



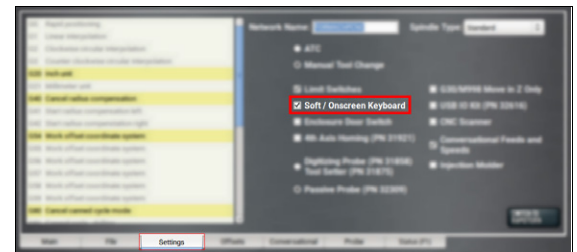
**Tip!** It's useful to program a G30 move right before a tool change so that the machine can jog to a safe tool change position.

### 5.2.6 Enable the On-Screen Keyboard

If you have an (optional) Touch Screen Kit (PN 35575), you can use a soft keyboard to type information in the PathPilot interface. For information, see "About Soft Keyboards" (on the next page).

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

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



**Figure 5-36: 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.



5: PATHPILOT TOOLS AND FEATURES
5.2 Machine Settings and Accessories

4. To close the keyboard, select the X key.



About Soft Keyboards

If you enabled a soft keyboard (on-screen keyboard) in the PathPilot interface to use with an optional touch screen or operator console, a keyboard opens when you select any field where keyboard input is required. The keyboard displays a wide range of keys: both uppercase and lowercase, symbols, arrow keys, caps lock, backspace and delete, and more.



Figure 5-37: Soft (on-screen) keyboard.

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

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

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

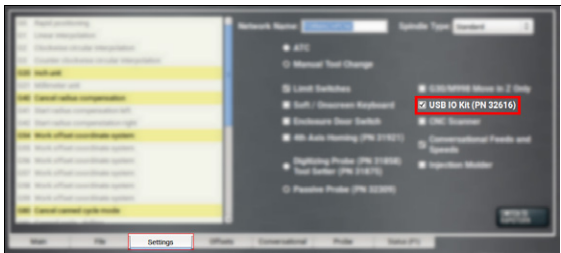


Figure 5-38: Settings tab.

5.2.8 Enable Tooltips

PathPilot displays expandable tooltips for many areas of the interface. Hovering over an item, like a DRO field or a button, displays helpful information about the item.

To enable or disable tooltips:

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

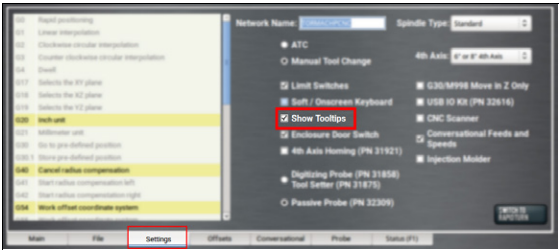


Figure 5-39: Show Tooltips checkbox.

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.

5.2.9 Enable Feeds and Speeds Suggestions in Conversational Routines

You can use PathPilot to automatically calculate feeds and speeds. For more information, see "Use Feeds and Speeds Suggestions" (page 98).

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

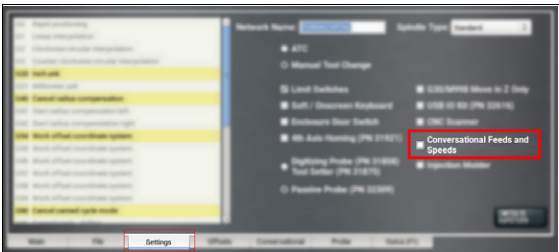


Figure 5-40: Settings tab.

5.2.10 Use a USB Camera

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



**Figure 5-41: USB camera status.**

As cameras are plugged in and unplugged, the **Camera Status** area is refreshed. To test compatibility of any USB camera, plug it in and watch the **Camera Status** area for the camera name and details.



**Note:** If a camera isn't shown after plugging it in or starting a video recording, it might require too much power from the USB ports on the controller. This is very likely when more than one camera is used. Try using a powered USB hub to add the camera(s).

When a USB camera is plugged in, it's analyzed for supported video and audio formats, frame sizes, and frame rates. If the camera supports it, PathPilot uses H.264 compression; otherwise, it uses Motion JPEG.

If the USB camera has a microphone, PathPilot records audio as well as video. The preferred format is compressed AAC, but uncompressed PCM is used as a fallback.

### About USB Cameras

Recording video and audio from USB cameras is supported in PathPilot v2.10.0 and later. You can use up to four cameras simultaneously to record from different vantage points.



**Note:** All cameras are started and stopped at the same time — if you don't want a camera to record, you must unplug it.

USB cameras are compatible with all machine types, but older controllers may lack the processing power and memory needed for camera support. Controllers require 4GB of memory for camera functionality. Use the ADMIN MEMORY MDI command to verify the memory size of a controller.

You can purchase a Tormach USB Camera (PN 51240) with a metal case, mounting bracket, and 15-foot USB cable. Other USB cameras may work (see below), but do not include any technical support.

## Manual Recording

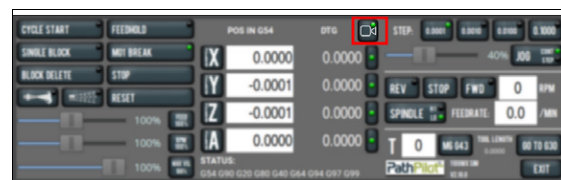
To start or stop a manual recording, either:

- Use the controls in the **Manual Recording** area of the **Camera(s)** tab.
- When a manual recording is stopped, a file save-as dialog appears prompting you for the file name base to use.



**Figure 5-42: Manual recording controls.**

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



**Figure 5-43: 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.

5: PATHPILOT TOOLS AND FEATURES

5.2 Machine Settings and Accessories



Figure 5-44: Camera settings.

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

1. Select **Snapshot** in the **Manual Recording** area of the **Camera(s)** tab.  
The **Main** tab displays.
2. Review the camera images, which display on top of the **Tool Path** area. The camera images refresh every 0.5 seconds.
3. Align the cameras or adjust lighting to your preference, and then select the **Shutter** button.

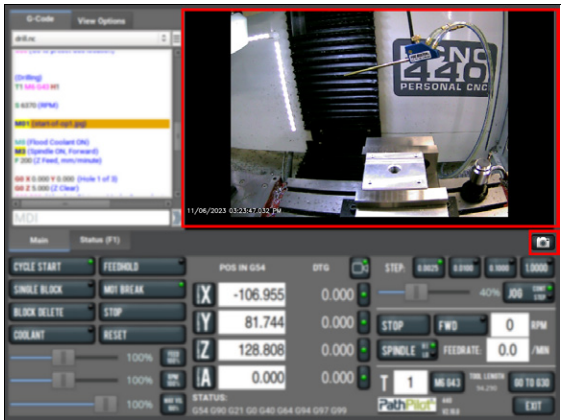


Figure 5-45: Example of taking a photo.

Automatic E-Stop Loop Recording ("Dashcam")

E-stop loop recording enables analysis of the previous 30 seconds after an E-stop. When enabled, recording is automatically started after reset.

To enable or disable the recording of emergency stops:

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



**Note:** This feature is enabled by default.

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

To view E-stop videos:

1. A slight delay occurs after an E-stop while the video is saved to the **E-stop Videos** folder. Select the video file, and then select **Load G-Code** to view it.

**Note:** The E-Stop Videos folder is automatically monitored for internal drive space use. If the folder size grows beyond 5 GB, the oldest video files are automatically deleted until the folder size becomes less than 5 GB.

Review Video and Image Files

1. On the **File** tab, select the video or image file and select **Load G-Code**.  
A video player application starts or the image preview is displayed.  
Alternatively, you could transfer the video or image files to a Windows or macOS computer for review.

File Naming Convention

For manual and automatic E-stop recordings, the base file name for the recording has automatically chosen suffixes appended for each camera.  
For example, if you stop a manual recording of two cameras, specify "Left Bracket Op1" as the name, and enabled screen recording, you'll see the following files:

| File Name               | Description of File                       |
|-------------------------|-------------------------------------------|
| Left Bracket Op1_0.mp4  | Camera 0 mp4 video file                   |
| Left Bracket Op1_0.log  | Troubleshooting log for camera 0          |
| Left Bracket Op1_1.mp4  | Camera 1 mp4 video file                   |
| Left Bracket Op1_1.log  | Troubleshooting log for camera 1          |
| Left Bracket Op1_PP.mp4 | PathPilot screen recording mp4 video file |
| Left Bracket Op1_PP.log | Troubleshooting log for screen recording  |

### G-Code Commands

PathPilot supports three new M-codes to control cameras within G-code programs: M301, M302, and M303. Example use cases:

- Record only across each M01 stop where the operator needs to flip a workpiece or change a tool.
- Create short videos that focus on unique aspects of the program to reduce later video editing.
- Record USB IO integration operations with robots or other devices (pneumatic vises, etc.).
- Monitor progress on a workpiece by including M303 throughout the program.

### File Naming Conventions

Recordings or pictures created by M301/M302/M303 have automatically generated file names, with the base file name taken from the running G-code file. Video files are saved alongside the G-code file. The suffix for each file uses a time stamp format. This makes it easy to distinguish multiple runs of the same G-code program.

For example, if engrave.nc is running and uses M301 and M302 to create one recording on a machine with one camera, and screen recording is enabled, you'll see the following files:

| File Name                          | Description of File                                 |
|------------------------------------|-----------------------------------------------------|
| engrave_2023-02-21_16_58_33_0.mp4  | Camera 0 mp4 video file                             |
| engrave_2023-02-21_16_58_33_0.log  | Troubleshooting log for camera 0                    |
| engrave_2023-02-21_16_58_33_PP.mp4 | PathPilot screen recording mp4 video file           |
| engrave_2023-02-21_16_58_33_PP.log | Troubleshooting log for screen recording            |
| engrave_2023-02-21_17_43_22.jpg    | Picture taken by a single M303 later in the program |

### Use M01 to Take Pictures

In addition to displaying information like pictures or messages during an M01 break, you can also use a USB camera (if installed) to take a picture.

To use M01 to take pictures:

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 147).

5: PATHPILOT TOOLS AND FEATURES

5.3 Set Up G-Code Programs

5.3 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.

|                                         |    |
|-----------------------------------------|----|
| 5.3.1 Use a Probe with PathPilot .....  | 82 |
| 5.3.2 Set Tool Length Offsets .....     | 84 |
| 5.3.3 Set Work Offsets .....            | 86 |
| 5.3.4 View Work Offsets .....           | 88 |
| 5.3.5 View Available G-Code Modes ..... | 88 |

5.3.1 Use a Probe with PathPilot

Use the Probe tab in the PathPilot interface to automate functions with a probe.

Set Up the Probe

Before using the functions on the Probe tab, you must first do the following:

- 1. Verify that tool number 99 (the probe tool) is in the spindle.
- 2. Disable the spindle to prevent any accidental spindle starts with the probe in the spindle.
- 3. Verify that the feed rate is appropriate for probing moves.

**Note:** All probing moves occur at a feed rate specified by the DRO fields on the Probe Setup tab.

- 4. Press the probe tip and make sure that, from the PathPilot interface, on the Probe tab, the Accessory Input light comes on.  
This indicates that the probe polarity is correctly specified.  
If the Accessory Input light does not come on, you must change the probe polarity setting.

Use a Probe to Find a Feature's Location

To find the location of a workpiece or vise in the current work offset coordinates:

- 1. From the PathPilot interface, on the **Probe** tab, select the **X/Y/Z Probe** tab.
- 2. Position the probe near the workpiece or vise.
- 3. One at a time, select **Find X+**, **Find X-**, **Find Y+**, **Find Y-**, or **Find Z-**.

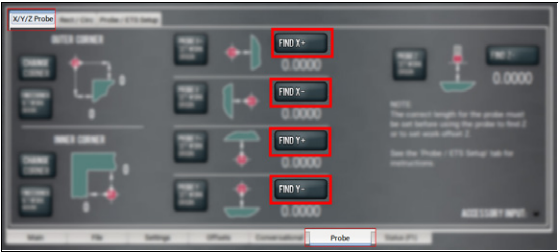


Figure 5-46: Probe tab.

The axis is probed, and the location of the probed surface is displayed.

Use a Probe to Set Work Offset Zeroes

You can set the work offsets of a workpiece or vise jaw using a probe.

Set the X and Y Work Offset Zero on the Corner of a Feature

- 1. From the PathPilot interface, on the **Probe** tab, select the **X/Y/Z Probe** tab.
- 2. Position the probe so that it is below the surface of the feature and 1 in. away from the vice jaw corner in the **X** and **Y** directions.
- 3. Select **Find Corner, Set Work Origin**.

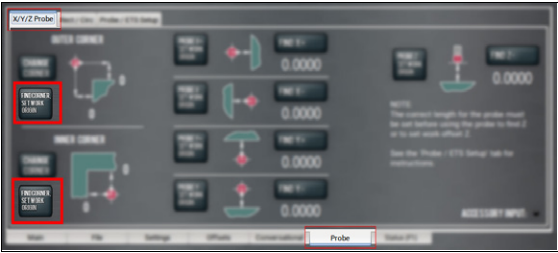


Figure 5-47: Probe tab.

The axes are probed, and the location of the probed surface is set as the current work offset's X/Y origin.

**Note:** Select **Change Corner** to change the corner on which to probe.

Set the Work Offset Zeroes on a Feature

- 1. From the PathPilot interface, on the **Probe** tab, select the **X/Y/Z Probe** tab.
- 2. Position the probe near the workpiece or vise.

- One at a time, select **Probe X+**, **Set Work Origin**, **Probe X-**, **Set Work Origin**, **Probe Y+**, **Set Work Origin**, **Probe Y-**, **Set Work Origin**, or **Probe Z-**, **Set Work Origin**.

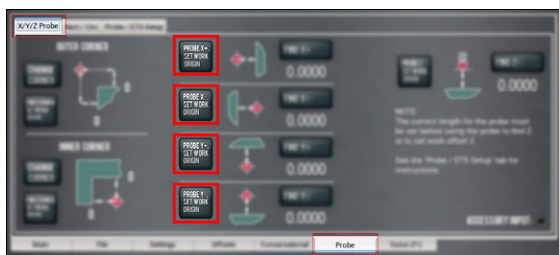


Figure 5-48: Probe tab.

The axis is probed, and the location of the probed surface is set as the current work offset's origin.

### Use a Probe to Find the Center of a Feature

You can find the center of a pocket, slot, or boss on a part using a probe.

#### Find the Center of a Pocket

- From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
- Position the probe near the center of the pocket.
- Select **Find Center, Set Work Origin** as shown in the following image.



Figure 5-49: Probe tab.

#### Find the Center of a Slot

- From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
- Position the probe near the center of the slot.
- Depending on the slot, do one of the following:
  - To probe the slot in the **X** direction only, select **Find Center, Set Work Origin** as shown in the following image.

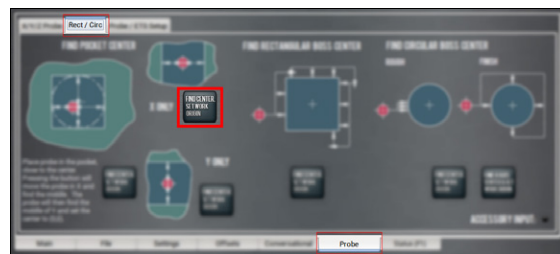


Figure 5-50: Probe tab.

- To probe the slot in the **Y** direction only, select **Find Center, Set Work Origin** as shown in the following image.



Figure 5-51: Probe tab.

#### Find the Center of a Rectangular Boss

- From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
- Position the probe below the top surface of the boss and on the left-hand side.
- Select **Find Center, Set Work Origin** as shown in the following image.



Figure 5-52: Probe tab.

The probe moves around the edge of the workpiece to find the center.

#### Find the Center of a Circular Boss

- From the PathPilot interface, on the **Probe** tab, select the **Rect/Circ** tab.
- Position the probe below the top surface of the boss and on the left-hand side.

5: PATHPILOT TOOLS AND FEATURES
5.3 Set Up G-Code Programs

- 3. Select Find Center, Set Work Origin as shown in the following image.

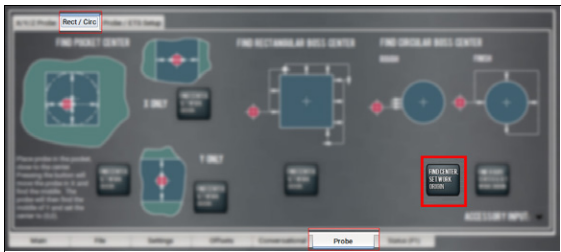


Figure 5-53: Probe tab.

The probe moves around the workpiece three times to determine the approximate center of the curve, and then makes four additional move to confirm the center of the circle.

Find the Center Rotation of an A-Axis

- 1. From the PathPilot interface, on the Probe tab, select the Rect/Circ tab.
- 2. Position the probe directly above the A-axis center of rotation.
- 3. Select Find A Axis Center & Set Work Origin.

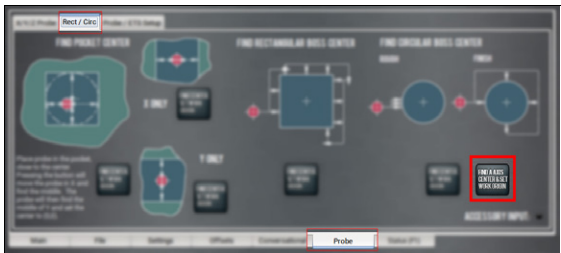


Figure 5-54: Probe tab.

The probe moves around the round workpiece mounted in the A-axis to find the center rotation of the A-axis.

5.3.2 Set Tool Length Offsets

Before running a G-code program, PathPilot must know the length of the tools that are required for the program. For more information on using tool length offsets, see "About Tool Offsets" (page 115).

Note: You can import a .csv file with tool length offset data. For information, see "Import and Export the Tool Table" (page 105).

To set tool length offsets:

- 1. Verify that the machine is powered on and out of reset.
- 2. Put a tool into a tool holder, and set it aside to measure.

- 3. From the PathPilot interface, on the Offsets tab, verify that the Tool tab is selected.
- 4. Find the Tool Table window.

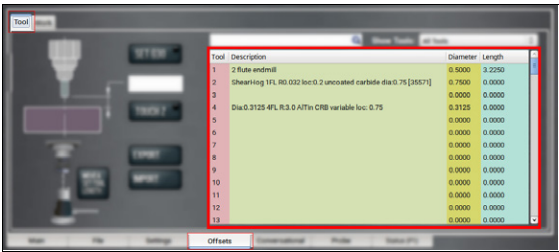


Figure 5-55: Tool Table window on the Offsets tab.

- 5. Depending on your workflow, you can measure your tools using any of the following methods:
• Use an Electronic Tool Setter For information, see Use an to Measure Tools.

About Tool Offsets

Tool offsets allow you to use various tools while still programming with respect to the workpiece. Tools can have different lengths .

The most common tool offset is the tool length offset: when you change tools, PathPilot must account for the difference in tool length. In CNC machines, the tool length offset is applied using a G43 command.

Before you begin a G-code program, you must verify the lengths of the tools in the program, and make sure that the lengths agree with the tool length offsets set in PathPilot:

- Each time you change tools, you must apply a new tool length offset in PathPilot.
- Each time you replace a tool, you must remeasure its length, and apply a new tool length offset in PathPilot.

NOTICE! You must always verify that the physical length of a tool agrees with the tool length offset value set in PathPilot. If you don't, there's a risk that the tool length offset misrepresents the currently active tool in the spindle, which may result in a machine crash or damaged tooling, workpieces, or fixtures.

Measure Tools Using an Electronic Tool Setter (ETS)

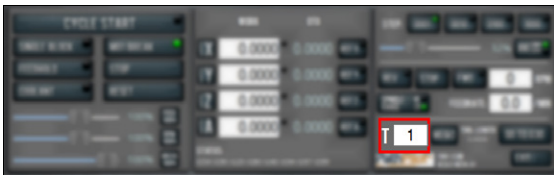
Depending on your workflow, do one of the following:

- Manually Measure Tool Lengths (with PathPilot). . . . . 85
- Automatically Measure Tool Lengths (with G37). . . . . 85

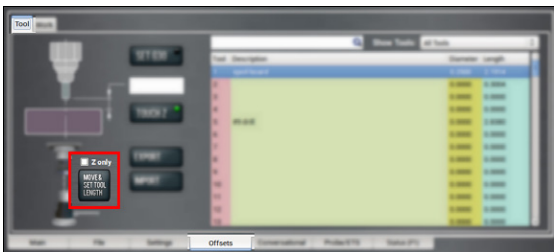
|                                                     |           |
|-----------------------------------------------------|-----------|
| <b>Set up the Electronic Tool Setter (ETS).....</b> | <b>85</b> |
| Set the ETS Height.....                             | 86        |
| Reference the Spindle Nose.....                     | 86        |
| Set the G37 Position.....                           | 86        |

### Manually Measure Tool Lengths (with PathPilot)

1. Set up the ETS.  
For information, see "Set up the Electronic Tool Setter (ETS)" (page 115).
2. From the PathPilot interface, on the **Offsets** tab, in the **Tool Table** window, in the **Description** column, type a description for the tool.
3. In the **Diameter** column, type the diameter of the tool. Then select the **Enter** key.
4. Install a tool into the spindle.
5. From the PathPilot interface, type the tool number in the **Tool DRO** field. Then select the **Enter** key.



6. On the **Offsets** tab, find the **Move and Set Tool Length** button and the **Z only** checkbox. Verify that the checkbox is cleared, and then select **Move and Set Tool Length**.



**Note:** You should only use the **Z only** checkbox to manually measure tools with a larger diameter. When it's selected, the machine doesn't go to the G37 position — instead, it just moves straight down (Z-) to measure the tool.

The machine moves to the G37 position and measures the tool with the ETS.



**Note:** Regardless of the initial feed rate, the final touch off feed rate while using an ETS is 2-1/2 in. per minute (IPM).

7. From the **Tool Table** window, in the **Length** column, verify that the length of the tool is correct.

### Automatically Measure Tool Lengths (with G37)

We recommend using the G37 G-code command to measure tools. This method simplifies the tool measurement procedure — you're letting the machine do the work for you — but it also increases tool length accuracy and reduces tool change times in multi-tool programs.

Depending on your workflow, do one of the following:

#### • Use G37 in the MDI Line DRO Field

1. Set up the ETS.  
For information, see "Set up the Electronic Tool Setter (ETS)" (page 115).
2. Put the ETS in its home position (that you determined in "Set up the Electronic Tool Setter (ETS)" (page 115)).
3. Put a tool into the spindle.
4. From the PathPilot interface, in the **MDI Line DRO** field, type G37. Then select the **Enter** key. The spindle moves to the ETS position, measures the length of the tool, and applies that length to the currently selected tool in the tool offsets table.

#### • Use G37 in a G-Code Program

1. Set up the ETS.  
For information, see "Set up the Electronic Tool Setter (ETS)" (page 115).
2. From the PathPilot interface, load a G-code program with a G37 command.
3. Select **Cycle Start**.  
When the program reaches the G37 command, the machine moves to the ETS and measures the length of the tool.

For information, see "Programming" (page 121).

### Set up the Electronic Tool Setter (ETS)

There are three steps to set up the ETS. Complete the following steps in the order listed:

|                                        |           |
|----------------------------------------|-----------|
| <b>Set the ETS Height.....</b>         | <b>86</b> |
| <b>Reference the Spindle Nose.....</b> | <b>86</b> |

5: PATHPILOT TOOLS AND FEATURES
5.3 Set Up G-Code Programs

Set the G37 Position 86

Set the ETS Height

Before you begin to use the ETS, you must first use the PathPilot interface to set its work offset.

To set the ETS height:

- 1. Jog the spindle until it's over the ETS.
- 2. From the PathPilot interface, set the active tool to 0.
- 3. From the PathPilot interface, on the Probe tab, select the ETS Setup tab. Then find the ETS Work Offset Setup group, and select Move & Set ETS Height.

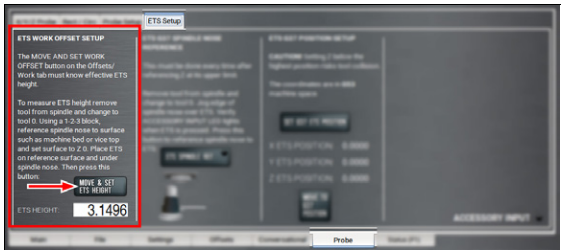


Figure 5-56: Move & Set ETS Height button on the Probe tab.

The Z-axis moves down (-Z) until the spindle nose contacts and triggers the ETS.

- 4. In the ETS Height DRO field, verify that the length of the ETS updated.

Reference the Spindle Nose

Note: You must repeat this procedure after each time that you reference the Z-axis.

- 1. Identify a home location for your ETS. You can use anywhere within the machine's area of travel as the home location, so long as you can center the spindle above the ETS.
- 2. Remove the tool holder from the spindle.
- 3. From the PathPilot interface, in the Tool DRO field, type 0. Then select the Enter key.
- 4. Jog the spindle until it's over the ETS. Then, slowly jog the Z-axis down (-Z) toward the contact pad on the ETS.
- 5. From the PathPilot interface, on the ETS Setup tab, find the ETS G37 Spindle Nose Reference section, and select ETS Spindle Ref.

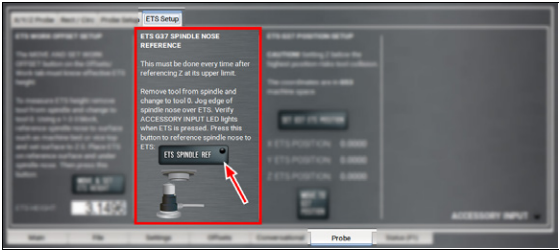


Figure 5-57: ETS Spindle Ref button on the Probe tab.

The Z-axis moves down (-Z) until the spindle nose contacts and triggers the ETS.

The spindle nose is now referenced to the ETS.

Set the G37 Position

- 1. Load the longest tool that you'll be using into the spindle. From the PathPilot interface, change to that tool.

NOTICE! If you use a shorter tool to set the G37 position, there's a risk of tool collision when using a longer tool.

- 2. Jog the tool so that it's about 20 mm above the tool setter's contact pad.
- 3. From the PathPilot interface, on the ETS Setup tab, find the ETS G37 Position Setup group, and select Set G37 ETS Position.

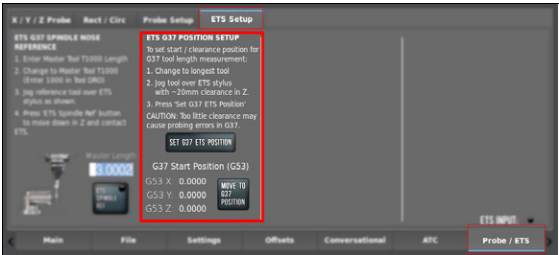


Figure 5-58: Set G37 ETS Position on the Probe tab.

The G37 position is now set.

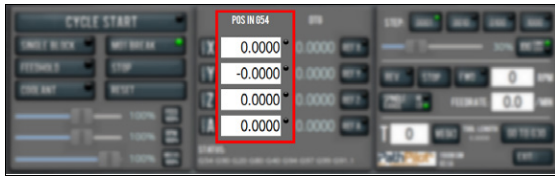
- 4. Verify that the X, Y, and Z ETS position displayed in their DRO fields are accurate.

Note: The values displayed in these DRO fields are in G53.

5.3.3 Set Work Offsets

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

- Select **Zero [Axis]**.



**Figure 5-59: Work Offset DRO fields.**

To change work offsets:

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



**Figure 5-60: Work offset indicated in the PathPilot interface.**

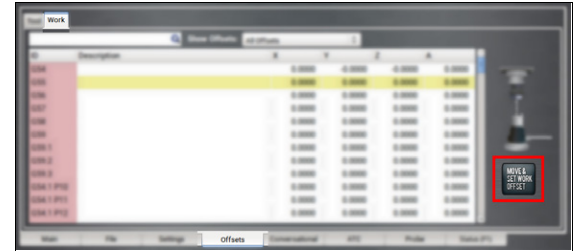
**Note:** The values in the **Work Offset** DRO fields update to indicate the new location of each axis in the new work offset.

For more information on using work offsets, see "About Work Offsets" (page 118).

### Set the Z-Axis Work Offset with an Electronic Tool Setter (ETS)

1. Set up the ETS.  
For information, see "Set up the Electronic Tool Setter (ETS)" (page 115).
2. Install a tool into the spindle.  
For information, see Install a Tool in an ER Collet Spindle.
3. Use the ETS to measure the length of the tool in the spindle.  
For information, see "Set Tool Length Offsets" (page 115).
4. Put the ETS on the surface that you want to set as Z zero.
5. Jog the spindle until the tool is centered over the ETS.

6. From the PathPilot interface, on the **Offsets** tab, on the **Work** tab, select **Move and Set Work Offset**.



**Figure 5-61: Work tab on the Offsets tab.**

The machine moves down (-Z) until the tool contacts the ETS. The Z-axis offset updates for the current work offset.

### 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 |

5: PATHPILOT TOOLS AND FEATURES

5.3 Set Up G-Code Programs

| Work Offset Naming |            |          |
|--------------------|------------|----------|
| 11                 | G54.1 P11  | Not used |
| ...                |            |          |
| 499                | G54.1 P499 | Not used |
| 500                | G54.1 P500 | Not used |

5.3.4 View Work Offsets

To view the current work offset:

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

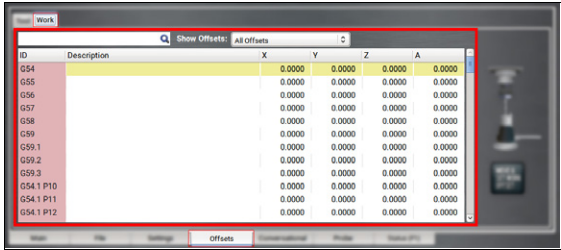


Figure 5-62: Work Offsets Table window.

The active work offset is highlighted.

To change the current work offset, go to "Set Work Offsets" (page 118).

5.3.5 View Available G-Code Modes

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

To view available G-code modes:

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

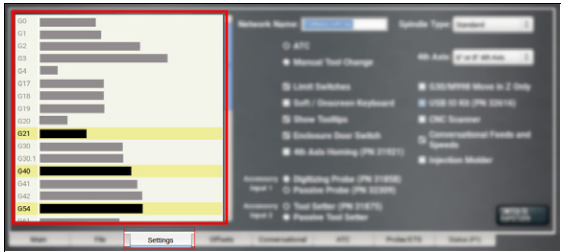


Figure 5-63: G-code Description window on the Settings tab.

## 5.4 RUN G-CODE PROGRAMS

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

|                                               |    |
|-----------------------------------------------|----|
| 5.4.1 Bring the Machine Out of Reset .....    | 89 |
| 5.4.2 View the Active Axis to Jog .....       | 89 |
| 5.4.3 Jog the Machine .....                   | 89 |
| 5.4.4 View the Current Machine Position ..... | 90 |
| 5.4.5 Reference the Machine .....             | 90 |
| 5.4.6 Start a Program .....                   | 91 |
| 5.4.7 Stop Machine Motion .....               | 91 |
| 5.4.8 Operate the Coolant Pump .....          | 91 |
| 5.4.9 View the Active G-Code Modes .....      | 92 |
| 5.4.10 View the Distance to Go .....          | 92 |

### 5.4.1 Bring the Machine Out of Reset

- Select **Reset**.

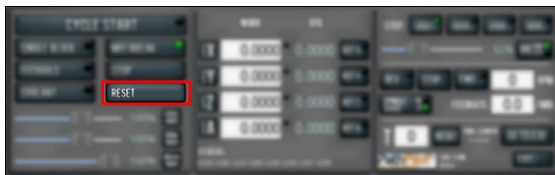


Figure 5-64: Reset button.

For more information on reset mode, see "About Reset Mode" (page 109).

#### 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.

### 5.4.2 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.

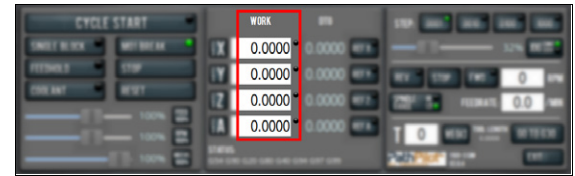


Figure 5-65: Work Offset DRO fields.

For information, see "Jog the Machine" (page 111).

### 5.4.3 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.

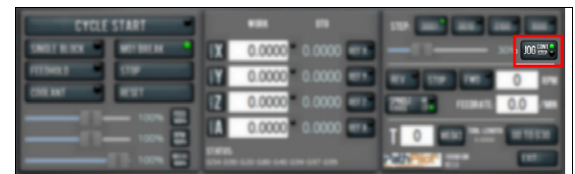


Figure 5-66: Jog button.

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.

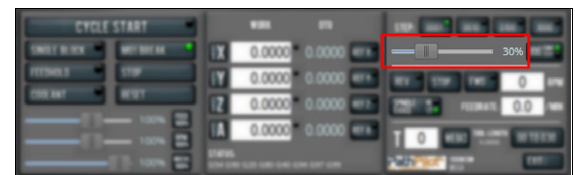


Figure 5-67: Jog Speed slider.

For more information on continuous velocity mode, see "About Continuous Velocity Jogging" (page 111).

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 5-68: Step buttons (in G20 mode).

# 5: PATHPILOT TOOLS AND FEATURES

## 5.4 Run G-Code Programs

- On the (optional) Jog Shuttle, press the Step button to toggle the currently selected step size.
- In the PathPilot interface, the **Step** button's light comes on, indicating which step size is active.

For more information on step mode, see "About Step Jogging" (page 111).

### 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 5-69: 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 5-70: Jog Speed slider.

### About Continuous Velocity Jogging

While jogging in continuous velocity mode, the machine moves at a constant speed for as long as:

- A keyboard key is pressed
- The Jog Shuttle outer ring is twisted away from the neutral position
- The Operator Console's pendant wheel is turned in the desired direction

This is useful when you're doing things like:

- Roughly positioning the machine (for example, to move the 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 5-71: Step buttons (in G20 mode).

### About Step Jogging

While jogging in step mode, the machine moves one step at a time. The jog step sizes range depending on the programming mode you are using:

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

Step jogging mode is useful to finely move the machine, like when you're indicating a workpiece or manually setting tool 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. The Operator Console's pendant moves the machine in the step size that's selected with the three-position switch.

### 5.4.4 View the Current Machine Position

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

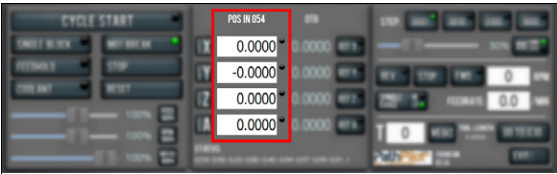


Figure 5-72: Work Offset DRO fields.

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 118).

### 5.4.5 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 5-73: Reference buttons.**

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



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

After each axis is referenced, its button light comes on. For more information on referencing the machine, see "About Referencing" (page 110).

#### 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.

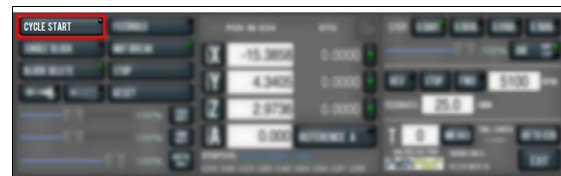
You must reference the machine at the following times:

- 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.

#### 5.4.6 Start a Program

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



**Figure 5-74: 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.

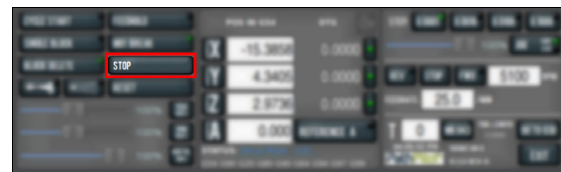
#### Cycle Start Reference

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 49).
- Before you've loaded a G-code program. For information, see "Load G-Code" (page 114).
- Before referencing the machine. For information, see "Reference the Machine" (page 110).

#### 5.4.7 Stop Machine Motion

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



**Figure 5-75: Stop button.**

#### 5.4.8 Operate the Coolant Pump

To turn coolant on or off:

# 5: PATHPILOT TOOLS AND FEATURES

## 5.4 Run G-Code Programs

- Select **Coolant**.

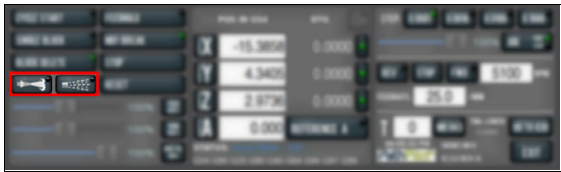


Figure 5-76: Coolant button.

For more information on turning on and off coolant, see "About Coolant" (page 120).

### About Coolant

In the PathPilot interface, the Coolant button controls the machine's coolant pump power outlet. The Coolant button's light shows the current state of the outlet: the light is on when the outlet has power; the light is off when the outlet does not have power.

**Note:** The Coolant button is equivalent to using an M08 (coolant on) or M09 (coolant off) command in the G-code program.

Use the Coolant button before or after a program is running, while a program is running, or while you are using manual data input (MDI) commands.

### 5.4.9 View the Active G-Code Modes

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

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



Figure 5-77: Status read-only DRO field.

For more information on G-code modes, go to "View Available G-Code Modes" (page 88).

### 5.4.10 View the Distance to Go

To view the distance to go:

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

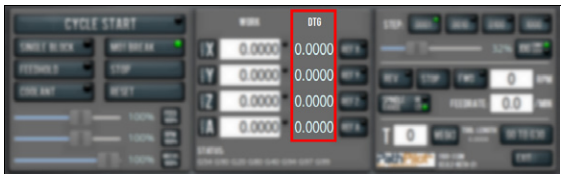


Figure 5-78: DTG read-only DRO fields.

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.

### 5.5 CONTROL G-CODE PROGRAMS

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

|                                                 |     |
|-------------------------------------------------|-----|
| 5.5.1 Use the Feed Hold Function .....          | 93  |
| 5.5.2 Use the Feed Rate Override Function ..... | 93  |
| 5.5.3 Use M01 Break Mode .....                  | 94  |
| 5.5.4 Use the Maxvel Override Function .....    | 94  |
| 5.5.5 Use Single Block Mode .....               | 95  |
| 5.5.6 Use the Spindle Override Function .....   | 95  |
| 5.5.7 Change the Feed Rate .....                | 96  |
| 5.5.8 Change the Spindle Speed .....            | 96  |
| 5.5.9 Change the Tool Number .....              | 96  |
| 5.5.10 Use a G30 Position .....                 | 96  |
| 5.5.11 View the Tool Length .....               | 97  |
| 5.5.12 Manually Enter Commands .....            | 97  |
| 5.5.13 Copy Recently Entered Commands .....     | 98  |
| 5.5.14 Use Feeds and Speeds Suggestions .....   | 98  |
| 5.5.15 Use Cycle Counters (M30 and M99) .....   | 101 |

#### 5.5.1 Use the Feed Hold Function

- Select **Feed Hold**.

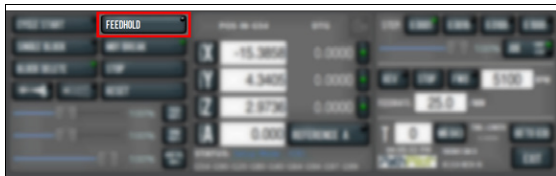


Figure 5-79: Feed Hold button.



**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 91).



**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.

#### 5.5.2 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 5-80: 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.

# 5: PATHPILOT TOOLS AND FEATURES

## 5.5 Control G-Code Programs

### 5.5.3 Use M01 Break Mode

- Select **M01 Break**.

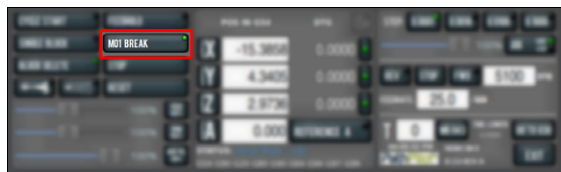


Figure 5-81: M01 Break button.

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 91).
- **When M01 Break is Inactive** PathPilot ignores all programmed M01 commands.

#### Display Information and Capture Images During an M00 or M01 Break

##### Display Information with Images

If the comment occurs on a line with **M00** or **M01**, and contains a file name with a .jpg or .png extension, PathPilot displays the image in the Tool Path display when the program reaches the M00 or M01 break.

To display an image during an M00 or M01 break:

1. Move an image file with a .jpg or .png extension to the PathPilot controller in one of the following locations:
  - In the same folder as the G-code program
  - In an images folder within the G-code program's folder
  - In an images folder within the home directory
2. Program an M00 or M01 break, and, using parentheses, type the full file name of the image (including its extension).



#### EXAMPLE

M01 (Op1\_Setup.jpg) displays the image file on the Tool Path display.

3. The image file displays on the Tool Path display.

#### Display Information with Text

To display a message on the Tool Path display:

1. Program an M00 or M01 break, and, using parentheses, type a message that you'd like to display on the screen.



#### EXAMPLE

M01 (Check coolant nozzles are pointed correctly) displays *Check coolant nozzles are pointed correctly* across the bottom of the Tool Path display.

2. The message displays on the Tool Path display.

#### Capture Images with a USB Camera

In addition to displaying information like pictures or messages during an M01 break, you can also use a USB camera (if installed) to take a picture.

To use M01 to take pictures:

1. Add **M01** (op1\_setup.jpg) into your G-code program.
2. Run the G-code program.
3. When PathPilot executes the M01 it looks to see if the comment contains a file name.
  - If there isn't a file name: The comment is shown as instructional text across the tool path.
  - If there is a file name, but the file doesn't exist yet and the extension is .jpg, .png, or .jpeg: The USB cameras are initialized and shown in the tool path display.
4. Select the **Shutter** button to take the picture and create the op1\_setup.jpg file.  
In future runs of the G-code program, **op1\_setup.jpg** will display to the operator for instructional purposes on the workpiece.

For more information, see "Use a USB Camera" (page 78).

### 5.5.4 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 5-82: 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%.

### 5.5.5 Use Single Block Mode

- Select **Single Block**.

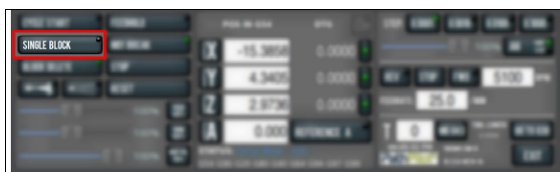


Figure 5-83: Single Block button.

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 91).

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.

### 5.5.6 Use the Spindle Override Function

To use the spindle override function:

- Using the **Spindle Override** slider, change the programmed spindle speed by a specific percentage.

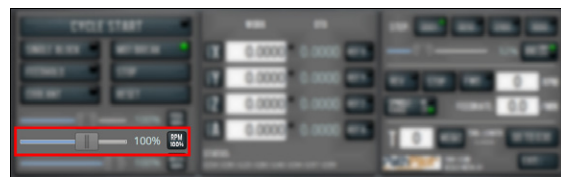


Figure 5-84: Spindle Override slider.



**Note:** Percentages range from 1-200%.

To remove the spindle override function:

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

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

### About Spindle Override

The spindle override function won't command the spindle to move past the maximum allowable speed. If the spindle isn't moving, the spindle override function is delayed until the next time spindle starts. The override doesn't drive the spindle past its maximum speed. It does affect the speed of a spindle command limited by a D word.

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

- Running a program
- Using manual data input (MDI) commands

The spindle override function is ignored in the following situations:

- If the program is running a spindle-synchronized move
- If an M48 (disable feed and speed overrides) command is used

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

# 5: PATHPILOT TOOLS AND FEATURES

## 5.5 Control G-Code Programs

### 5.5.7 Change the Feed Rate

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



Figure 5-85: Feed Rate DRO field.

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

#### About Feed Rates

A feed rate is the velocity at which the workpiece can be fed against the tool in the machine's spindle.

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

### 5.5.8 Change the Spindle Speed

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



Figure 5-86: Spindle RPM DRO field.

For information, see "About Spindle Controls" (below).

#### About Spindle Controls

A spindle speed is the rate at which the spindle rotates. Use the FWD and Stop buttons to manually control the spindle.

| Button | G-Code | Use to...                                                                      |
|--------|--------|--------------------------------------------------------------------------------|
| FWD    | M03    | Start the spindle clockwise at the RPM specified in the Spindle RPM DRO field. |
| Stop   | M05    | Stop the spindle.                                                              |

The FWD button and the Spindle RPM field don't operate if selected when:

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

### 5.5.9 Change the Tool Number

The Tool DRO field shows the tool number currently active.

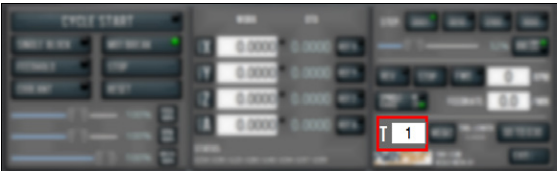


Figure 5-87: Tool DRO field.

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

1. In the **Tool** DRO field, type a number (the valid range is from 0-1000). Then select the **Enter** key.



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

#### About M6 G43

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

- Change the number of the currently-loaded tool in the spindle to the number typed in the Tool DRO field. This is the equivalent of an M06 command.
- Apply the tool length offset for that tool typed in the Tool DRO field. For more information on tool length offsets, see "Set Tool Length Offsets" (page 115). This is the equivalent of a G43 command.

### 5.5.10 Use a G30 Position

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

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

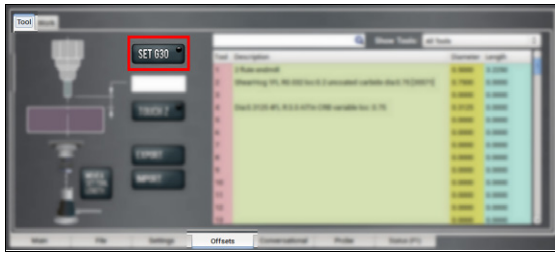


Figure 5-88: Set G30 button.

To go to a set G30 position:

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

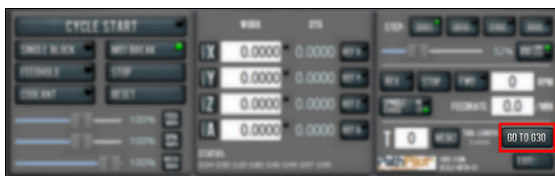


Figure 5-89: Go to G30 button.



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

### About G30

A G30 command in a G-code program moves the machine to a preset position. For more information on setting a G30 position, see "Use a G30 Position" (on the previous page). Use a G30 move to start a coordinated movement of the axes. You can limit the movement to only the Z-axis. For information, see "Limit G30 Moves" (page 77).



**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.

### 5.5.11 View the Tool Length

- Identify the **Tool Length** read-only DRO field.

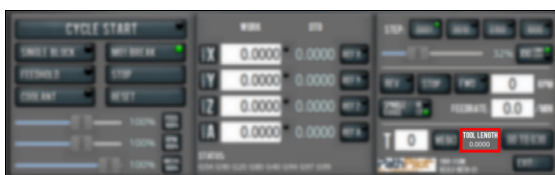


Figure 5-90: Tool Length DRO field.

If the tool offset matches the number of the tool in the **Tool DRO** field, the text is light blue on a gray background.

If the tool offset doesn't match the number of the tool in the **Tool DRO** field, the text is orange on a red background.

### 5.5.12 Manually Enter Commands

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

To manually enter commands:

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

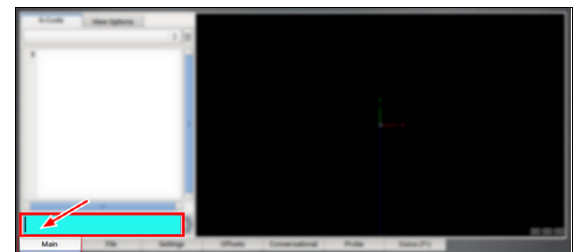


Figure 5-91: MDI Line DRO field.

The DRO field highlights.

2. Type the command.



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

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

### About the MDI Line DRO Field

The MDI Line DRO field allows you to send commands (or, manual data input) directly to PathPilot. For information, see "Manually Enter Commands" (above).

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

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

### Admin Commands Reference

Use the following commands in PathPilot:

| Admin Command | Use to...                                         |
|---------------|---------------------------------------------------|
| ADMIN AUDIO   | Customize the controller's audio device settings. |

## 5: PATHPILOT TOOLS AND FEATURES

### 5.5 Control G-Code Programs

| Admin Command           | Use to...                                                                               |
|-------------------------|-----------------------------------------------------------------------------------------|
| ADMIN CALC              | Open the calculator.                                                                    |
| ADMIN CLEAR             | Clear the message history on the Status tab.                                            |
| ADMIN CONFIG            | Change the configuration of the PathPilot interface.                                    |
| ADMIN CYCLECOUNTER      | Control the cycle counters in the Tool Path display.                                    |
| ADMIN DATE              | Customize the controller's date and time.                                               |
| ADMIN DISPLAY           | Customize the controller's screen display.                                              |
| ADMIN DROPBOX           | Connect your controller to a Dropbox account for cloud file syncing.                    |
| ADMIN HELP              | Review a list of available Admin commands.                                              |
| ADMIN KEYBOARD          | Customize the controller's keyboard layout.                                             |
| ADMIN LOGDATA           | Write the latest machine log data to a USB drive for technical support assistance.      |
| ADMIN MEMORY            | Determine how much total RAM is on your controller.                                     |
| ADMIN MOUSE             | Change the mouse preferences, like pointer speed and right- or left-hand button mapping |
| ADMIN NETWORK           | Configure a Wi-Fi network.                                                              |
| ADMIN RESET_SOFT_LIMITS | Reset axis soft limits to machine defaults.                                             |
| ADMIN SET_X_LIMIT       | Set the X-axis soft limit.                                                              |
| ADMIN SET_Y_LIMIT       | Set the Y-axis soft limit.                                                              |
| ADMIN SET_Z_LIMIT       | Set the Z-axis soft limit.                                                              |

| Admin Command               | Use to...                                                                                                                         |
|-----------------------------|-----------------------------------------------------------------------------------------------------------------------------------|
| 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 DELAYS        | 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.                                                                           |

#### 5.5.13 Copy Recently Entered Commands

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

For information, see "Manually Enter Commands" (on the previous page).

#### 5.5.14 Use Feeds and Speeds Suggestions



**Note:** Calculating feeds and speeds requires that PathPilot has relevant details about the tooling. If you haven't yet done so, go to "Create Tool Descriptions" (page 100).

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

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

## Conversational Feeds and Speeds.

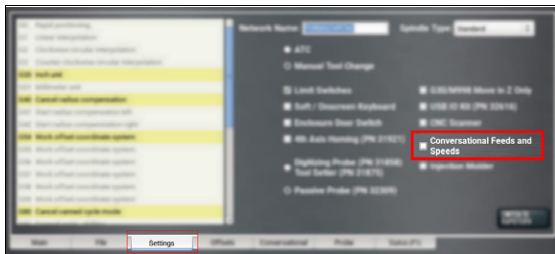


Figure 5-92: Settings tab.

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



Figure 5-93: Feeds and speeds suggestions on the Conversational tab.

- From the **Material** dropdown, select your material (like **Aluminum** or **Plastic**).
- If required, from the **Sub-Type** dropdown, select the material sub-type (like **-any-** or **6061**).
- In the **Tool** DRO field, type the assigned tool number.
- Select **Refresh** (to the right of the **Sub-Type** dropdown). The following machining-related DRO fields are calculated:

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



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

- (Optional) You can adjust the values in the calculated DRO fields. Adjusting the value in one of these DRO field doesn't change the value in the other machining-related DRO fields.



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

## Refresh DRO Field Values

The suggested values are no longer valid if:

- You select different material or sub-type values, or if you type a new value in to the Tool DRO field. The suggested feeds and speeds are made by taking into account all of these values. Changing any value requires you to refresh.
- You select a different Conversational tab. The suggested feeds and speeds are made by taking into account the current, specific conversational operation. Changing your conversational operation requires you to refresh the feeds and speeds values.

When the feeds and speeds are no longer valid, the Refresh button turns green, and the machining-related DRO field backgrounds switch from green to white, as shown in the following image.

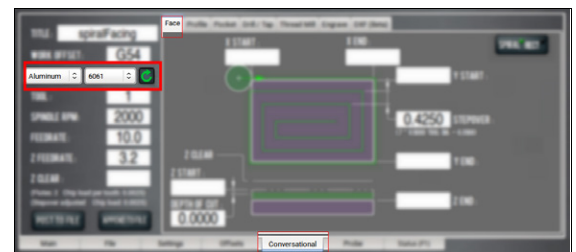


Figure 5-94: Refresh button on the Conversational tab.

## Use Additional Provided Information

The following tips are displayed based on the calculations that PathPilot is performing:

- **Chip Load Information** Chip load — the amount of material removed per tooth — is based on the number of flutes, RPM, and feed rate. Chip thinning takes the stepover (the horizontal depth of cut into the workpiece) into account, and provides the actual chip load. As the stepover value decreases, the actual chip load decreases. If the stepover is too small, the cutter may

5: PATHPILOT TOOLS AND FEATURES

5.5 Control G-Code Programs

- not have enough contact with the material to cut — effectively resulting in premature tool wear.
- **Cutting Speed Information** Cutting speed is the speed that a given tooth (flute) on the cutter will be moving when it cuts through the material. All materials have a documented cutting speed.  
In imperial units, cutting speed is measured as surface feet per minute (SFM).  
In metric units, cutting speed is measured as surface meters per minute (SMM).
  - **Material Removal Information** The material removal rate (MRR) indicates how much material is removed by the tool per minute while cutting.  
In imperial units, cutting speed is measured as cubic inches per minute.  
In metric units, cutting speed is measured as cubic centimeters per minute.

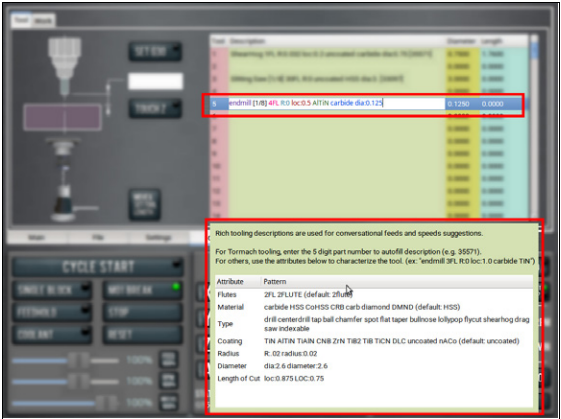


Figure 5-95: A manually-entered tool description.

Examples

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

 **EXAMPLE**  
Dia.:.3125 4FL R:0.03 ALTiN CRB variable loc:.75

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

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

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

 **EXAMPLE**  
Gold colored end mill from middle drawer

This description provides very little information, and PathPilot defaults to basic cutter features:

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

Automatically Generate Tool Descriptions

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

Create Tool Descriptions

If desired, you can create tool descriptions in PathPilot. Detailed tool descriptions allow you to receive feeds and speeds suggestions in conversational programming. For information, see "Use Feeds and Speeds Suggestions" (page 98).

Manually Enter Tool Descriptions

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

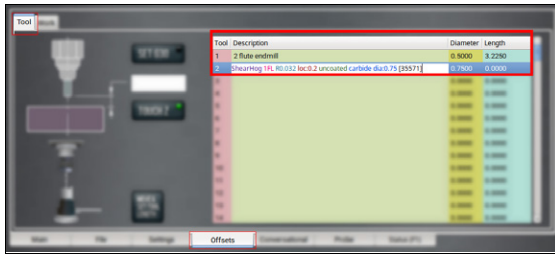
1. From the PathPilot interface, on the **Offsets** tab, identify the **Tool Table** window.
2. Select a blank line.
3. Type a description for the tool. Descriptions are not case sensitive.  
If a pattern or word in the description is recognized, PathPilot uses syntax highlighting to indicate a valid description.

window.



**Note:** If you don't know the part number, you can search for the tool at [tormach.com](http://tormach.com).

1. From the PathPilot interface, on the **Offsets** tab, in the **Tool Table** window, select a blank line.
2. Type the part number for the tool, like **35571**.  
The full description and tool diameter for a ShearHog (PN 35571) displays.



**Figure 5-96: An automatically-generated tool description for a Tormach tool.**

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

### Tool Keywords Reference

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

| Item   | Pattern                                                                                                                                                | Example                       | Notes                                                                                 |
|--------|--------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------|---------------------------------------------------------------------------------------|
| type   | drill,<br>centerdrill,<br>tap, ball,<br>chamfer,<br>spot, flat,<br>taper,<br>bullnose,<br>lollypop,<br>flycut,<br>shearhog,<br>drag, saw,<br>indexable | DRILL, BALL,<br>FLY CUT, DRAG | “Drag” indicates that the tool is a drag tool, and has no (0) RPM associated with it. |
| flutes | A number followed by “FL” or “FLUTE”                                                                                                                   | 4FL, 12FL,<br>2FLUTE          | No flutes is specified the same as two flutes.                                        |

| Item                            | Pattern                                                               | Example                        | Notes                                                                           |
|---------------------------------|-----------------------------------------------------------------------|--------------------------------|---------------------------------------------------------------------------------|
| length of cut (or flute length) | “loc” followed by a colon, followed by a decimal number               | LOC:0.875                      | If no length of cut is specified, a length is assumed based on cutter diameter. |
| tool coating                    | TiN, AlTiN, TiAlN, CNB, ZrN, TiB2, TiB, TiCN, DLC, uncoated, nACo     | TiN, ZrN, TiB2                 | No coating is specified same as “uncoated.”                                     |
| tool diameter                   | “diameter” or “dia” followed by a colon, followed by a decimal number | DIAMETER: .0341<br>, DIA: .750 | —                                                                               |
| tool material                   | carbide, HSS, CoHSS, CRB, carb, diamond, DMND                         | HSS, COHSS, CRB                | No tool material is specified the same as HSS (high-speed steel).               |
| tool radius                     | “R” or “radius” followed by a colon, followed by a decimal number     | R: .02,<br>RADIUS:0.02         | No radius is specified the same as a zero radius.                               |

### 5.5.15 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

# 5: PATHPILOT TOOLS AND FEATURES

## 5.5 Control G-Code Programs

unattended operation. For each M-code, there's an A and B counter. This provides more flexibility, because you can reset them to 0 independently. For example, you could use M30 A to count parts each shift, and M30 B to count parts each week. The cycle counters persist across the controller's power cycles.

### Monitor Cycle Counters

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

### Change Cycle Counter Values

The cycle counters are implemented as read-only persistent G-code numbered parameters, as detailed in the following table. If needed, the cycle counter value can be read in G-code.

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

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

`L99 P~ Q~`

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

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

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

## 5.6 SYSTEM FILE MANAGEMENT

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

- 5.6.1 Manage System Files .....103
- 5.6.2 Create Backup Files ..... 103
- 5.6.3 Restore Backup Files ..... 104
- 5.6.4 Import and Export the Tool Table ..... 105

### 5.6.1 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 58).

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**.

### 5.6.2 Create Backup Files

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

**Note:** To prevent errors when backing up and restoring files, only use a blank, formatted USB drive. Do not use the PathPilot v2.0 Upgrade.

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

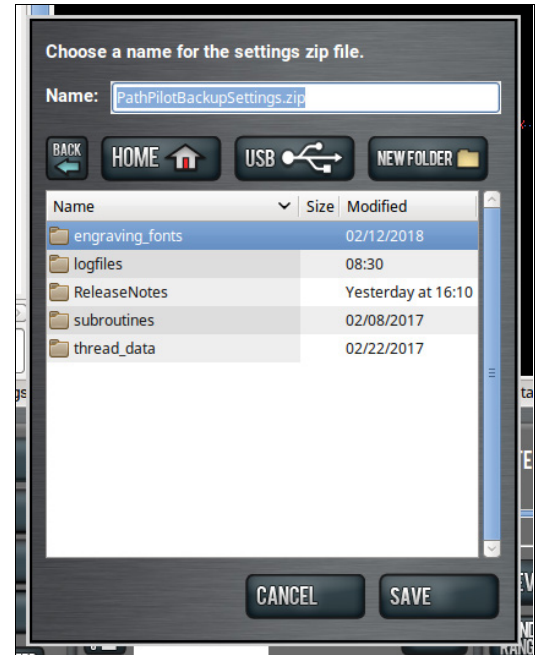


Figure 5-97: Admin Settings Backup dialog box.

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.

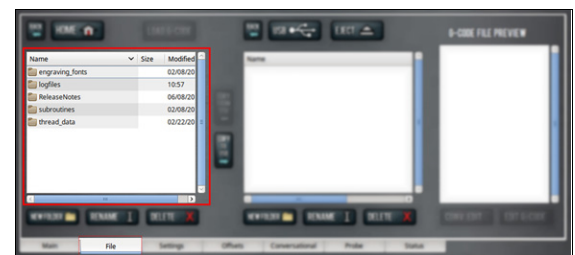


Figure 5-98: Controller Files window on the File tab.

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

# 5: PATHPILOT TOOLS AND FEATURES

## 5.6 System File Management

- To prevent errors, make sure you don't include the following folders:
  - logfiles
  - media
  - ReleaseNotes
  - subroutines
  - USB
- Select **Copy to USB**.  
The files are copied and display in the **USB Files** window.
- Eject the USB drive from the PathPilot controller.
- From the PathPilot interface, select **Exit**.
- Verify that all files are properly saved: insert the USB drive on a device other than the PathPilot controller, and review the list of files on the USB drive.
- (Optional) As an extra precaution, copy all the files onto the device.
- Go to Install PathPilot v2.x.

### About Backup Files

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

#### 5.6.3 Restore Backup Files

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

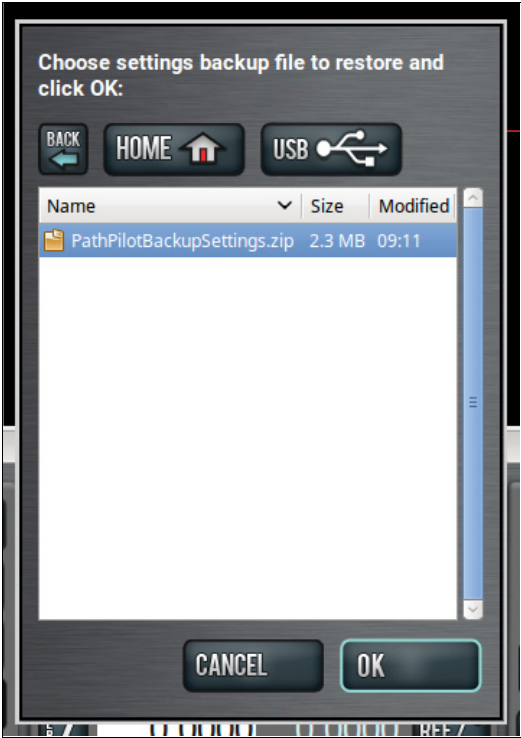


Figure 5-99: Admin Settings Restore dialog box.

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

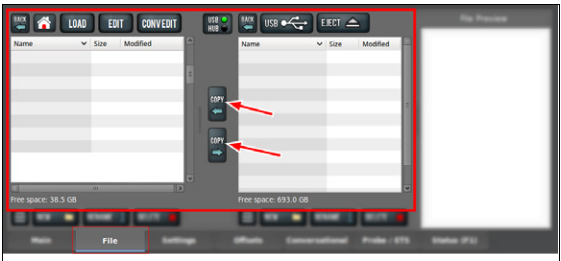


Figure 5-100: USB Files window on the File tab.

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

- From the **Controller Files** window, select the folder into which you want to copy the files.

6. Select **Copy From USB**.

The files display in the **Controller Files** window.

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

7. If desired, you can enable an Internet connection in the PathPilot controller, which allows you to use Dropbox with PathPilot, and receive automatic PathPilot updates. Go to the next section, "Enable an Internet Connection" (page 74).  
If you don't want to use these features, you have completed installing and configuring PathPilot v2.x.

### 5.6.4 Import and Export the Tool Table

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

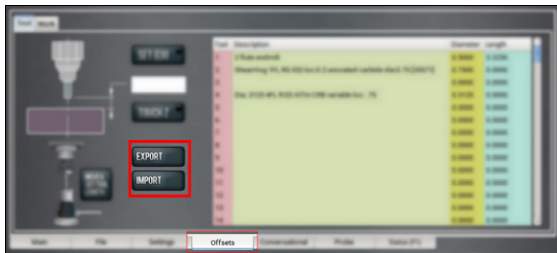


Figure 5-101: Export and Import buttons on the Offsets tab.

#### 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.

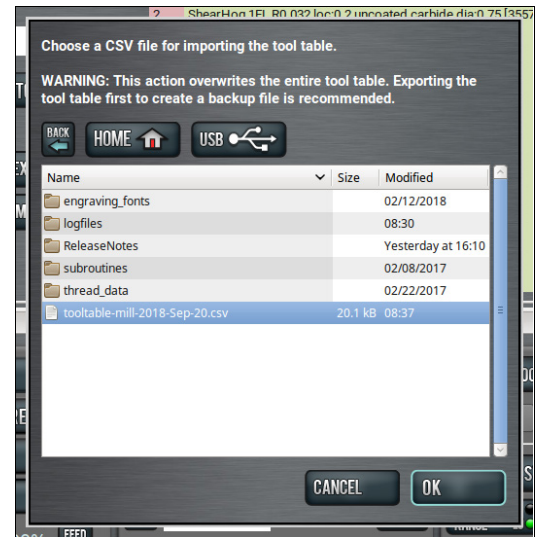


Figure 5-102: Import dialog box.

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.

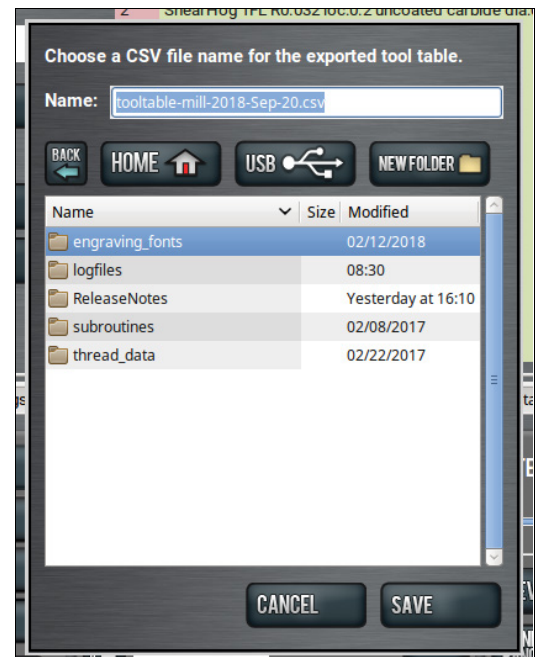


Figure 5-103: Export dialog box.

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.

## 5: PATHPILOT TOOLS AND FEATURES

### 5.6 System File Management

4. From the **File** tab, select the newly created .csv file, and then select **Copy to USB**.
5. Select **Eject**.  
It's safe to remove the USB drive from the controller.

# BASIC OPERATIONS

## IN THIS SECTION, YOU'LL LEARN:

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

---

### CONTENTS

|                                         |     |
|-----------------------------------------|-----|
| 6.1 Start the Machine.....              | 108 |
| 6.2 Bring the Machine Out of Reset..... | 109 |
| 6.3 Reference the Machine.....          | 110 |
| 6.4 Jog the Machine.....                | 111 |
| 6.5 Manually Control the Spindle.....   | 113 |
| 6.6 Load G-Code.....                    | 114 |
| 6.7 Set Tool Length Offsets.....        | 115 |
| 6.8 Set Work Offsets.....               | 118 |
| 6.9 Operate the Coolant Pump.....       | 120 |

## 6: BASIC OPERATIONS

### 6.1 Start the Machine

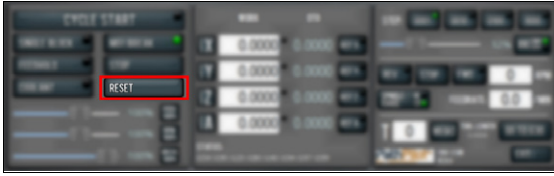
#### 6.1 START THE MACHINE

Power on the machine and the PathPilot controller.

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

#### 6.2 BRING THE MACHINE OUT OF RESET

- Select **Reset**.



**Figure 6-1: Reset button.**

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

##### 6.2.1 About Reset Mode

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

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

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

## 6: BASIC OPERATIONS

### 6.3 Reference the Machine

#### 6.3 REFERENCE THE MACHINE

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



**Figure 6-2: Reference buttons.**

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



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

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

#### 6.3.1 About Referencing

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

You must reference the machine at the following times:

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

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

## 6.4 JOG THE MACHINE

To switch between jogging modes:

- From the **Manual Control** area, in the **Jog** group, select **Jog**.  
PathPilot toggles between continuous velocity mode and step mode.



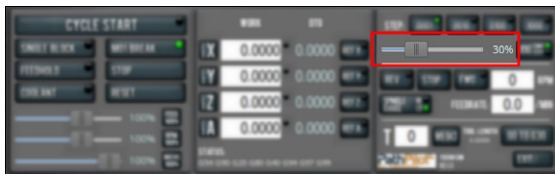
**Figure 6-3: Jog button.**

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

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

To use continuous velocity mode:

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

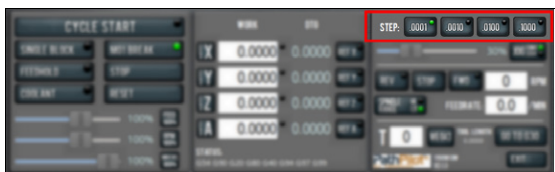


**Figure 6-4: Jog Speed slider.**

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

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

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



**Figure 6-5: Step buttons (in G20 mode).**

- On the (optional) Jog Shuttle, press the Step button to toggle the currently selected step size.  
In the PathPilot interface, the **Step** button's light comes on, indicating which step size is active.

For more information on step mode, see "About Step Jogging" (below).

### 6.4.1 About Jogging

Jogging is the operation of manually moving an axis in various directions (like to set up and indicate fixtures or workpieces). You can't manually jog the machine while it's performing automatic operations (like running a G-code program or an MDI command).

Jog the machine using the keyboard, Jog Shuttle, or Operator Console pendant. Whichever device you're jogging with, you can either:

- Jog the machine at a consistent velocity (for information, see "About Continuous Velocity Jogging" (below)).
- Jog the machine in steps (for information, see "About Step Jogging" (below)).

For more information, see "Jog Controls Reference" (on the next page).

#### About Continuous Velocity Jogging

While jogging in continuous velocity mode, the machine moves at a constant speed for as long as:

- A keyboard key is pressed
- The Jog Shuttle outer ring is twisted away from the neutral position
- The Operator Console's pendant wheel is turned in the desired direction

This is useful when you're doing things like:

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

#### About Step Jogging

While jogging in step mode, the machine moves one step at a time. The jog step sizes range depending on the programming mode you are using:

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

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

# 6: BASIC OPERATIONS

## 6.4 Jog the Machine

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

### 6.4.2 Jog Controls Reference

The machine's jogging functions are controlled by the following:

- The Jog group of the Manual Control area in the PathPilot interface
- The keyboard
- The (optional) Jog Shuttle or Operator Console pendant

| Axis   | Direction | Keyboard Key | Jog Shuttle      |
|--------|-----------|--------------|------------------|
| X-Axis | Positive  | Right Arrow  | Clockwise        |
|        | Negative  | Left Arrow   | Counterclockwise |
| Z-Axis | Positive  | Page Up      | Clockwise        |
|        | Negative  | Page Down    | Counterclockwise |
| A-Axis | Positive  | Period       | Clockwise        |
|        | Negative  | Comma        | Counterclockwise |

#### Jogging in PathPilot

From the PathPilot interface, in the Manual Control area, the Jog group has the following functions:

- The Jog button, which toggles between continuous velocity mode and step mode.
- The Jog Speed slider, which controls the machine's jog rate (whether in continuous velocity mode or in step mode).  
The jog rate is measured as a percentage of the machine's maximum jog rate.

#### Jogging with the Keyboard

Pressing the keys results in the following actions:

- **[KEY]** jogs the axis at the current jog rate.
- **[KEY]+Shift** jogs the axis at the maximum jog rate.

#### Jogging with the (Optional) Jog Shuttle or Operator Console Pendant

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

- **Axis Movement**
  - Jog Shuttle: An inner wheel provides precise, incremental jogging. Each detent (click) moves the

selected axis by one jog step increment. An outer ring provides smooth, continuous jogging based on rotation speed.

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



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

- **Axis Selection**

- Jog Shuttle: Four buttons let you toggle between the X, Y, Z, and A axes.
- Pendant: A four-position switch selects the active axis.



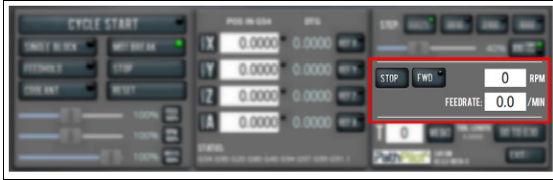
**Note:** The currently selected axis displays in PathPilot: in the Position Status group, there's a green light to the left of the Axis DRO field. When it's on, it indicates the active axis.

- **Step Size Selection**

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

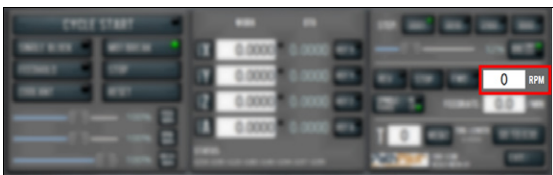
#### 6.5 MANUALLY CONTROL THE SPINDLE

1. Verify that the machine is powered on and out of reset.
2. From the PathPilot interface, in the **Manual Control** area, locate the **Spindle** group.



**Figure 6-6: Spindle group.**

3. In the **RPM DRO** field, type the desired RPM speed. Then select the **Enter** key.



**Figure 6-7: RPM DRO field.**

4. Select **FWD** to start the spindle in the forward direction.
5. Select **Stop** to stop the spindle.

##### 6.5.1 About the Spindle

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

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

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

# 6: BASIC OPERATIONS

## 6.6 Load G-Code

### 6.6 LOAD G-CODE

To run a G-code program on a PathPilot controller, you must first verify that the file is on the controller. For more information on transferring and moving files, see "Transfer Files to and From the Controller" (page 58).

To load G-code:

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

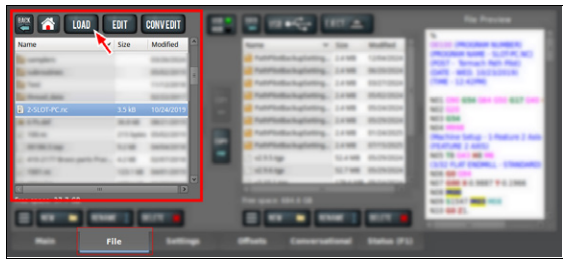


Figure 6-8: Controller Files window on the File tab.



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

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

### 6.7 SET TOOL LENGTH OFFSETS

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

**Note:** You can import a .csv file with tool length offset data. For information, see "Import and Export the Tool Table" (page 105).

To set tool length offsets:

1. Verify that the machine is powered on and out of reset.
2. Put a tool into a tool holder, and set it aside to measure.
3. From the PathPilot interface, on the **Offsets** tab, verify that the **Tool** tab is selected.
4. Find the **Tool Table** window.

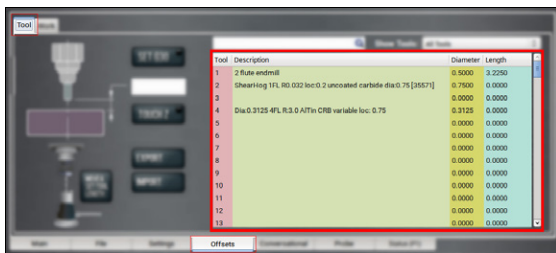


Figure 6-9: Tool Table window on the Offsets tab.

5. Depending on your workflow, you can measure your tools using any of the following methods:
  - **Use an Electronic Tool Setter** For information, see Use an to Measure Tools.

#### 6.7.1 About Tool Offsets

Tool offsets allow you to use various tools while still programming with respect to the workpiece. Tools can have different lengths .

The most common tool offset is the tool length offset: when you change tools, PathPilot must account for the difference in tool length. In CNC machines, the tool length offset is applied using a G43 command.

Before you begin a G-code program, you must verify the lengths of the tools in the program, and make sure that the lengths agree with the tool length offsets set in PathPilot:

- Each time you change tools, you must apply a new tool length offset in PathPilot.
- Each time you replace a tool, you must remeasure its length, and apply a new tool length offset in PathPilot.

**NOTICE!** You must always verify that the physical length of a tool agrees with the tool length offset value set in PathPilot. If you don't, there's a risk that the tool length offset misrepresents the currently active tool in the spindle, which may result in a machine crash or damaged tooling, workpieces, or fixtures.

### Set up the Electronic Tool Setter (ETS)

There are three steps to set up the ETS. Complete the following steps in the order listed:

- Set the ETS Height ..... 115
- Reference the Spindle Nose ..... 115
- Set the G37 Position ..... 116

#### Set the ETS Height

Before you begin to use the ETS, you must first use the PathPilot interface to set its work offset.

To set the ETS height:

1. Jog the spindle until it's over the ETS.
2. From the PathPilot interface, set the active tool to 0.
3. From the PathPilot interface, on the **Probe** tab, select the **ETS Setup** tab. Then find the **ETS Work Offset Setup** group, and select **Move & Set ETS Height**.

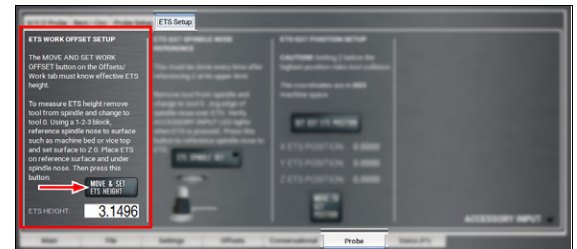


Figure 6-10: Move & Set ETS Height button on the Probe tab.

The Z-axis moves down (-Z) until the spindle nose contacts and triggers the ETS.

4. In the **ETS Height** DRO field, verify that the length of the ETS updated.

#### Reference the Spindle Nose



**Note:** You must repeat this procedure after each time that you reference the Z-axis.

6: BASIC OPERATIONS
6.7 Set Tool Length Offsets

- 1. Identify a home location for your ETS. You can use anywhere within the machine's area of travel as the home location, so long as you can center the spindle above the ETS.
- 2. Remove the tool holder from the spindle.
- 3. From the PathPilot interface, in the Tool DRO field, type 0. Then select the Enter key.
- 4. Jog the spindle until it's over the ETS. Then, slowly jog the Z-axis down (-Z) toward the contact pad on the ETS.
- 5. From the PathPilot interface, on the ETS Setup tab, find the ETS G37 Spindle Nose Reference section, and select ETS Spindle Ref.

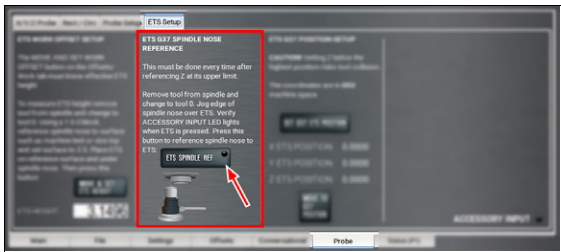


Figure 6-11: ETS Spindle Ref button on the Probe tab.
The Z-axis moves down (-Z) until the spindle nose contacts and triggers the ETS.
The spindle nose is now referenced to the ETS.

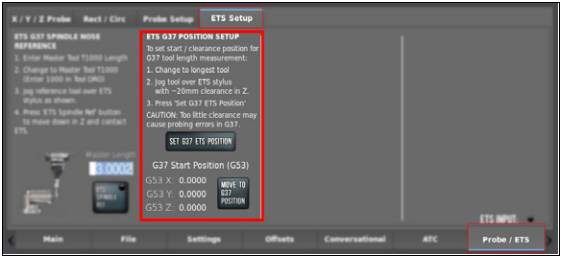


Figure 6-12: Set G37 ETS Position on the Probe tab.
The G37 position is now set.

- 4. Verify that the X, Y, and Z ETS position displayed in their DRO fields are accurate.

Note: The values displayed in these DRO fields are in G53.

Measure Tools Using an Electronic Tool Setter (ETS)

Depending on your workflow, do one of the following:

- Manually Measure Tool Lengths (with PathPilot). . . . . 116
- Automatically Measure Tool Lengths (with G37). . . . . 117

Manually Measure Tool Lengths (with PathPilot)

- 1. Set up the ETS.
For information, see "Set up the Electronic Tool Setter (ETS)" (on the previous page).
- 2. From the PathPilot interface, on the Offsets tab, in the Tool Table window, in the Description column, type a description for the tool.
- 3. In the Diameter column, type the diameter of the tool. Then select the Enter key.
- 4. Install a tool into the spindle.
- 5. From the PathPilot interface, type the tool number in the Tool DRO field. Then select the Enter key.



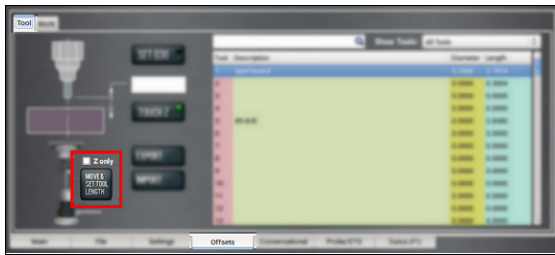
Set the G37 Position

- 1. Load the longest tool that you'll be using into the spindle. From the PathPilot interface, change to that tool.

NOTICE! If you use a shorter tool to set the G37 position, there's a risk of tool collision when using a longer tool.

- 2. Jog the tool so that it's about 20 mm above the tool setter's contact pad.
- 3. From the PathPilot interface, on the ETS Setup tab, find the ETS G37 Position Setup group, and select Set G37 ETS Position.

- On the **Offsets** tab, find the **Move and Set Tool Length** button and the **Z only** checkbox. Verify that the checkbox is cleared, and then select **Move and Set Tool Length**.



**Note:** You should only use the **Z only** checkbox to manually measure tools with a larger diameter. When it's selected, the machine doesn't go to the G37 position — instead, it just moves straight down (Z-) to measure the tool.

The machine moves to the G37 position and measures the tool with the ETS.

**Note:** Regardless of the initial feed rate, the final touch off feed rate while using an ETS is 2-1/2 in. per minute (IPM).

- From the **Tool Table** window, in the **Length** column, verify that the length of the tool is correct.

### Automatically Measure Tool Lengths (with G37)

We recommend using the G37 G-code command to measure tools. This method simplifies the tool measurement procedure — you're letting the machine do the work for you — but it also increases tool length accuracy and reduces tool change times in multi-tool programs.

Depending on your workflow, do one of the following:

#### • Use G37 in the MDI Line DRO Field

- Set up the ETS.  
For information, see "Set up the Electronic Tool Setter (ETS)" (page 115).
- Put the ETS in its home position (that you determined in "Set up the Electronic Tool Setter (ETS)" (page 115)).
- Put a tool into the spindle.
- From the PathPilot interface, in the **MDI Line DRO** field, type G37. Then select the **Enter** key.

The spindle moves to the ETS position, measures the length of the tool, and applies that length to the currently selected tool in the tool offsets table.

#### • Use G37 in a G-Code Program

- Set up the ETS.  
For information, see "Set up the Electronic Tool Setter (ETS)" (page 115).
- From the PathPilot interface, load a G-code program with a G37 command.
- Select **Cycle Start**.  
When the program reaches the G37 command, the machine moves to the ETS and measures the length of the tool.

For information, see "Programming" (page 121).

6: BASIC OPERATIONS

6.8 Set Work Offsets

6.8 SET WORK OFFSETS

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

- Select **Zero [Axis]**.

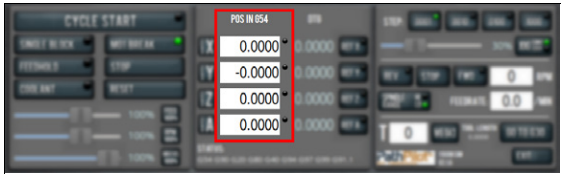


Figure 6-13: Work Offset DRO fields.

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 6-14: 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).

6.8.1 Set the Z-Axis Work Offset with an Electronic Tool Setter (ETS)

1. Set up the ETS.  
For information, see "Set up the Electronic Tool Setter (ETS)" (page 115).
2. Install a tool into the spindle.  
For information, see Install a Tool in an ER Collet Spindle.
3. Use the ETS to measure the length of the tool in the spindle.

For information, see "Set Tool Length Offsets" (page 115).

4. Put the ETS on the surface that you want to set as Z zero.
5. Jog the spindle until the tool is centered over the ETS.
6. From the PathPilot interface, on the **Offsets** tab, on the **Work** tab, select **Move and Set Work Offset**.

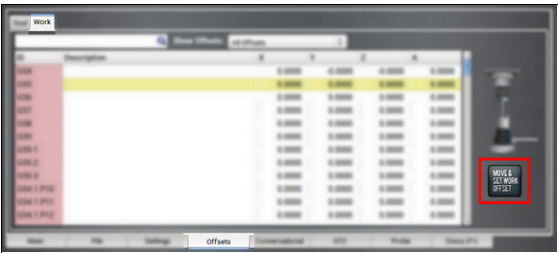


Figure 6-15: Work tab on the Offsets tab.

The machine moves down (-Z) until the tool contacts the ETS. The Z-axis offset updates for the current work offset.

6.8.2 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 |

| Work Offset Naming                 |               |          |
|------------------------------------|---------------|----------|
| 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 |

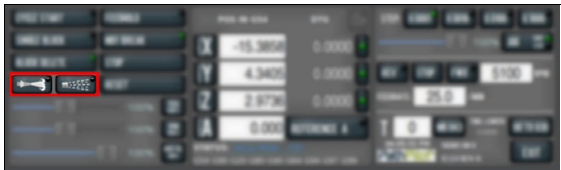
# 6: BASIC OPERATIONS

## 6.9 Operate the Coolant Pump

### 6.9 OPERATE THE COOLANT PUMP

To turn coolant on or off:

- Select **Coolant**.



**Figure 6-16: Coolant button.**

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

#### 6.9.1 About Coolant

In the PathPilot interface, the Coolant button controls the machine's coolant pump power outlet. The Coolant button's light shows the current state of the outlet: the light is on when the outlet has power; the light is off when the outlet does not have power.

 **Note:** The Coolant button is equivalent to using an M08 (coolant on) or M09 (coolant off) command in the G-code program.

Use the Coolant button before or after a program is running, while a program is running, or while you are using manual data input (MDI) commands.

# PROGRAMMING

## IN THIS SECTION, YOU'LL LEARN:

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

---

### CONTENTS

|                                     |     |
|-------------------------------------|-----|
| 7.1 Before You Begin .....          | 122 |
| 7.2 Programming Overview .....      | 123 |
| 7.3 Programming G-Code .....        | 129 |
| 7.4 Programming Canned Cycles ..... | 141 |
| 7.5 Programming M-Code .....        | 146 |
| 7.6 Programming Input Codes .....   | 150 |
| 7.7 Advanced Programming .....      | 151 |

# 7: PROGRAMMING

## 7.1 Before You Begin

### 7.1 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.

### 7.2 PROGRAMMING OVERVIEW

Read the following sections for a G-code overview:

|                                                      |            |
|------------------------------------------------------|------------|
| <b>7.2.1 About G-Code Programming Language</b> ..... | <b>123</b> |
| <b>7.2.2 G-Code Formatting Reference</b> .....       | <b>123</b> |
| <b>7.2.3 Supported G-Codes Reference</b> .....       | <b>127</b> |

#### 7.2.1 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).

#### 7.2.2 G-Code Formatting Reference

A permissible block of input code is made up of the following programming elements, in order, with the restriction that there is a maximum of 256 characters allowed on a line:

1. (Optional) Block delete character (/)
2. (Optional) Line number

3. Any number of words, parameter settings, and comments
4. End of line marker (carriage return or line break)

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



#### EXAMPLE

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

A line may have:

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

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

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

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

#### Line Numbers

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

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

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

#### Words

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

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

# 7: PROGRAMMING

## 7.2 Programming Overview

### Letters

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



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

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

| Letter | Description |
|--------|-------------|
| X      | X-axis      |
| Y      | Y-axis      |
| Z      | Z-axis      |

### Values

A real value is one of the following:

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



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

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

Explicit numbers have the following rules (in this case, a digit is a single character, 0 through 9):

- A number must consist of the following, in the order listed:
  1. An optional plus or minus sign
  2. Zero to many digits
  3. (Optional) One decimal point
  4. Zero to many digits
- There must be at least one digit somewhere in the number.
- It must be either an integer (no decimal point) or a decimals (decimal point).
- It may have any number of digits (subject to line length limitations).



**Note:** PathPilot only keeps 17 significant figures, which is enough for all known applications.

- A non-zero number with no sign as the first character is assumed to be positive.

Initial zeros (a zero before the decimal point and the first non-zero digit) and trailing zeros (a zero after the decimal point and the last non-zero digit) are allowed, but not required. A number written with initial or trailing zeros has the same value when it is read as if the extra zeros were not there. Numbers used for specific purposes by PathPilot are often restricted to some finite set of values, or to some range of values. In many uses, decimal numbers must be close enough to an integer to be accepted as a valid input. A decimal number which is supposed to be close to an integer is considered close enough if it is within 0.0001 of an integer.

### Order of Execution

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

The order of items on a line doesn't determine the order of execution on the commands.

Three types of items' order may vary on a line (as given earlier in this section):

- **Word** May be reordered in any way without changing the meaning of the line.
- **Parameter Setting** If it's reordered, there is no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter takes effect.

#### EXAMPLE

When the line #3=15 #3=6 is interpreted, the value of parameter 3 is 6. If the order is reversed to #3=6 #3=15 and the line is interpreted, the value of parameter 3 is 15.

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



#### EXAMPLE

G40 G01 #3=15 (foo) #4=-7.0 has five items and means exactly the same thing in any of the 120 possible orders, like #4=-7.0 G01 #3=15 G40 (foo), for the five items.

The order of execution of items on a line is critical to safe and effective machine operation. If items occur on the same line, they are executed in a particular order. To impose a different order (like to turn coolant off before the spindle is stopped), code the commands on separate blocks.

The order of execution is as follows:

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



**Note:** This is not supported.

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

# 7: PROGRAMMING

## 7.2 Programming Overview

modified by G53)

23. Stop (M00, M01, M02, M30, M60)

### Modal Groups

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

#### EXAMPLE

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

A few G-codes and M-codes are non-modal (like Dwell (G04)). These codes have effect only on the lines on which they occur. Modal commands are arranged in sets, called modal groups. Only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time (like inch units (G20) vs. millimeter units (G21)).

A machining system may be in many modes at the same time, with one mode from each modal group being in effect. For all G-code modal groups, when a machining system is ready to accept commands, one member of the modal group must be in effect. There are default settings for these modal groups. When the machining system is turned on or re-initialized, default values are automatically in effect.

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

| Group   | Commands                                                                                 | Group Description             |
|---------|------------------------------------------------------------------------------------------|-------------------------------|
| Group 1 | {G00, G01, G02, G03, G33, G37/G37.1, G38.x, G73, G76, G80, G81, G82, G85, G86, G88, G89} | Motion (one always in effect) |
| Group 2 | {G17, G18, G19, G17.1, G17.2, G17.3}                                                     | Plane selection               |

| Group    | Commands                                            | Group Description                    |
|----------|-----------------------------------------------------|--------------------------------------|
| Group 3  | {G90, G91}                                          | Distance mode                        |
| Group 4  | {G90.1, G91.1}                                      | Arc distance mode                    |
| Group 5  | {G93, G94}                                          | Feed rate mode                       |
| Group 6  | {G20, G21}                                          | Length units                         |
| Group 7  | {G40, G41, G42, G41.1, G42.1}                       | Cutter compensation                  |
| Group 8  | {G43, G43.1, G49}                                   | Tool length offset                   |
| Group 10 | {G98, G99}                                          | Return mode in canned cycles         |
| Group 12 | {G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3} | Select work offset coordinate system |
| Group 13 | {G61, G61.1, G64}                                   | Path control mode                    |
| Group 14 | {G96, G97}                                          | Spindle control mode                 |
| Group 15 | {G07, G08}                                          | Lathe diameter mode                  |

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

| Group   | Commands                  | Group Description                                                          |
|---------|---------------------------|----------------------------------------------------------------------------|
| Group 4 | {M00, M01, M02, M30, M60} | Program stop and program end                                               |
| Group 7 | {M03, M05}                | Spindle control                                                            |
| Group 8 | {M07, M08, M09}           | Coolant control (special case: M07 and M08 may be active at the same time) |

| Group   | Commands   | Group Description |
|---------|------------|-------------------|
| Group 9 | {M48, M49} | Override control  |

Non-modal G-codes are:

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

### Comments

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



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

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



### EXAMPLE :

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

### 7.2.3 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                   |
| <u>G17, G18, G19</u> | Plane selection                         |
| <u>G20/G21</u>       | Length units                            |

| G-Code              | Description                                    |
|---------------------|------------------------------------------------|
| <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>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                                   |
| <u>G86</u>          | Boring cycle                                   |
| <u>G88</u>          | Boring cycle                                   |
| <u>G89</u>          | Boring cycle                                   |
| <u>G90, G90.1</u>   | Arc distance mode                              |

7: PROGRAMMING

7.2 Programming Overview

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

### 7.3 PROGRAMMING G-CODE

Read the following sections as a G-code reference:

|                                                                                    |            |
|------------------------------------------------------------------------------------|------------|
| <b>7.3.1 About the Examples Used</b> .....                                         | <b>129</b> |
| <b>7.3.2 Rapid Linear Motion (G00)</b> .....                                       | <b>129</b> |
| <b>7.3.3 Linear Motion at Feed Rate (G01)</b> .....                                | <b>129</b> |
| <b>7.3.4 Arc at Feed Rate (G02 and G03)</b> .....                                  | <b>130</b> |
| <b>7.3.5 Dwell (G04)</b> .....                                                     | <b>132</b> |
| <b>7.3.6 Set Offsets (G10)</b> .....                                               | <b>132</b> |
| <b>7.3.7 Plane Selection (G17, G18, G19)</b> .....                                 | <b>133</b> |
| <b>7.3.8 Length Units (G20 and G21)</b> .....                                      | <b>133</b> |
| <b>7.3.9 Return to Predefined Position (G28 and G28.1)</b> ....                    | <b>133</b> |
| <b>7.3.10 Return to Predefined Position (G30 and G30.1)</b> ....                   | <b>134</b> |
| <b>7.3.11 Automatically Measure Tool Lengths with an ETS (G37 and G37.1)</b> ..... | <b>134</b> |
| <b>7.3.12 Straight Probe (G38.x)</b> .....                                         | <b>135</b> |
| <b>7.3.13 Cutter Compensation (G40, G41, G42)</b> .....                            | <b>136</b> |
| <b>7.3.14 Dynamic Cutter Compensation (G41.1 and G42.1)</b> .....                  | <b>136</b> |
| <b>7.3.15 Apply Tool Length Offset (G43)</b> .....                                 | <b>137</b> |
| <b>7.3.16 Engrave Sequential Serial Number (G47)</b> .....                         | <b>137</b> |
| <b>7.3.17 Cancel Tool Length Compensation (G49)</b> .....                          | <b>137</b> |
| <b>7.3.18 Absolute Coordinates (G53)</b> .....                                     | <b>137</b> |
| <b>7.3.19 Select Work Offset Coordinate System (G54 to G54.1 P500)</b> .....       | <b>138</b> |
| <b>7.3.20 Set Exact Path Control Mode (G61)</b> .....                              | <b>138</b> |
| <b>7.3.21 Set Blended Path Control Mode (G64)</b> .....                            | <b>138</b> |
| <b>7.3.22 Distance Mode (G90 and G91)</b> .....                                    | <b>138</b> |
| <b>7.3.23 Arc Distance Mode (G90.1 and G91.1)</b> .....                            | <b>138</b> |
| <b>7.3.24 Temporary Work Offsets (G92, G92.1, G92.2, and G92.3)</b> .....          | <b>138</b> |
| <b>7.3.25 Feed Rate Mode (G93, G94, and G95)</b> .....                             | <b>139</b> |
| <b>7.3.26 Spindle Control Mode (G96 and G97)</b> .....                             | <b>139</b> |

#### 7.3.1 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.

#### 7.3.2 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

#### 7.3.3 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.

# 7: PROGRAMMING

## 7.3 Programming G-Code

### 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

#### 7.3.4 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 depending on arc distance mode (G90.1/G91.1) of the center of the circle (Y and Z directions, respectively).

It's an error if:

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



### EXAMPLE

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

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

```
(Sample Program G02EX3:)
(Workpiece Size: X4, Y3, Z1)
(Tool: Tool #2, 1/4" Slot Drill)
(Tool Start Position: X0, Y0, Z1)
N2 G90 G80 G40 G54 G20 G17 G94 G64 (SAFETY BLOCK)
N5 G90 G20
```

7: PROGRAMMING
7.3 Programming G-Code

```
N10 M06 T2 G43 H2
N15 M03 S1200
N20 G00 X1 Y1
N25 Z0.1
N30 G01 Z-0.1 F5
N35 G02 X2 Y2 I1 J0 F20 (ARC FEED CW, RADIUS I1,J0
AT 20 IPM)
N40 G01 X3.5
N45 G02 X3 Y0.5 R2 (ARC FEED CW, RADIUS 2)
N50 X1 Y1 R2 (ARC FEED CW, RADIUS 2)
N55 G00 Z0.1
N60 X2 Y1.5
N65 G01 Z-0.25
N70 G02 X2 Y1.5 I0.25 J-0.25 (FULL CIRCLE ARC FEED
MOVE CW)
N75 G00 Z1
N80 X0 Y0
N85 M05
N90 M30
```

7.3.5 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

7.3.6 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).....132
Set Coordinate System (G10 L2).....132
Set Tool Table (G10 L10).....132
Set Tool Table (G10 L11).....133
Set Coordinate System (G10 L20).....133

Set Tool Table (G10 L1)

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

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

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

Troubleshooting

It's an error if:

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

Set Coordinate System (G10 L2)

To define the origin of a work offset coordinate system, program: G10 L2 P~ <axes R~>

- P~ is the number of coordinate system to use (G54 = 1, G59.3 = 9)
• R~ is the rotation about the rotation about the Z-axis

The G10 L2 P~ command doesn't change from the current coordinate system to the one specified by P. Use G54 through G59.3 to select a coordinate system.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed. If it's currently active, the new coordinates take effect immediately. For example, if a G92 origin offset was in effect before G10 L2, it continues to be in effect after.

Optionally program R to indicate the rotation of the XY axis around the Z-axis. The direction of rotation is counterclockwise (viewed from the positive end of the Z-axis). When a rotation is in effect, jogging an axis only moves that axis in a positive or a negative direction — not along the rotated axis. To cancel a rotation for the active coordinate, program G10 L2 P0 R0.

Troubleshooting

It's an error if:

- The P number does not evaluate to an integer in the range 0-500
• An axis other than X or Z is programmed

Set Tool Table (G10 L10)

To change the tool table entry for tool P so that if the tool offset is reloaded with the machine in its current position and with the current G5x and G92 offsets active, program: G10 L10 P~ R~

- $P\sim$  is the tool number
- $R\sim$  is the radius of tool

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

### Troubleshooting

It's an error if:

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

### Set Tool Table (`G10 L11`)

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

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

- $P\sim$  is the tool number
- $R\sim$  is the radius of tool

### Troubleshooting

It's an error if:

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

### Set Coordinate System (`G10 L20`)

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

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

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

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

### Troubleshooting

It's an error if:

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

### 7.3.7 Plane Selection (`G17`, `G18`, `G19`)

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

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

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

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

### 7.3.8 Length Units (`G20` and `G21`)

To set length units to inches, program: `G20`

To set length units to millimeters, program: `G21`



**Tip!** Program either `G20` or `G21` near the beginning of a program, before any motion occurs. Avoid using either one anywhere else in the program. It's your responsibility to make sure that all numbers are appropriate for use with the current length units.

### 7.3.9 Return to Predefined Position (`G28` and `G28.1`)

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

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



**Note:** Any axis not specified won't move.

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

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

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

## 7: PROGRAMMING

### 7.3 Programming G-Code

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

#### Troubleshooting

It's an error if:

- Cutter Compensation is turned on

#### 7.3.10 Return to Predefined Position (G30 and G30.1)

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

To make a rapid traverse move from the current position to the absolute position of the values in parameters, program: `G30`

To make a rapid traverse move to the position specified by axes including any offsets, then make a rapid traverse move to the absolute position of the values in parameters 5181 and/or 5183, program: `G30 X~ Z~`



**Note:** Any axis not specified won't move.

To store the current absolute position into parameters 5181-5183, program: `G30 .1`

#### Troubleshooting

It's an error if:

- Cutter Compensation is turned on

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

Use G37 and G37.1 with an Electronic Tool Setter (ETS) to enable automatic length measurement. For automated use, add a G37 command after an M6 tool change commands.

If you're using the ETS with a mill, the input port varies depending on your machine:

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

#### Move to G37 Position Over ETS (G37.1)

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

To set the G37 position:

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

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

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

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



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

G37.1 performs as follows:

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



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

#### Move and Measure Tool Length (G37)

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

- `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 a symmetrical 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 (positive or negative), the G-code program stops.  
For example, **G37 P0.005** passes if the measured length is within  $\pm 0.005$  of the current tool table length.
- **E~** overrides the pullout tolerance with the specified value.  
Use **E~** together with **P~** to set separate thresholds for tool breakage (**P~**) and tool pullout (**E~**), allowing asymmetric tolerance checking.

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

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

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

**G37** performs as follows:

1. Issues a **G37.1** move to the ETS location.
2. A downward rough probe feed rate move until the tool triggers the ETS.
3. An upward retract move of 0.100 in. to back off the triggered ETS.
4. A downward slow ETS probe feed rate move until the tool triggers the ETS.
5. An upward retract move of 0.100 in. to back off the triggered ETS.
6. An upward rapid move to the **G37 ETS Z** position.

### 7.3.12 Straight Probe (G38.x)

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

**G38.3** probes toward the workpiece and stops on contact

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

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

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

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

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

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

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

### Troubleshooting

It's an error if:

- The current point is less than 0.01 in. (0.254 mm) from the programmed point
- **G38** is used in inverse time feed rate mode
- Any rotational axis is commanded to move
- No X-, Y- or Z-axis word is used  
The linear axis words are optional, except that at least one of them must be used.
- Feed rate is zero
- The probe is already tripped

### Use the Straight Probe Command

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

- Without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane
- If the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane

## 7: PROGRAMMING

### 7.3 Programming G-Code

- 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> endsup
M02
```

#### 7.3.13 Cutter Compensation (G40, G41, G42)

To turn Cutter Compensation off, program: G40

It's okay to turn compensation off when it is already off.

It's an error if:

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

To program Cutter Compensation to the left of the programmed tool path (as viewed looking down on the machine), program: G41 D~

To program Cutter Compensation to the right of the programmed tool path (as viewed looking down on the machine), program: G42 D~

- D~ is the tool number associated with the diameter offset to be applied

The D word is optional — if there is no D word, the radius of the currently loaded tool is used. If no tool is loaded and no D word is given, a radius of 0 is used. If supplied, the D word is the tool number to use.

The lead in move must be at least as long as the tool radius.

The lead in move can be a rapid move.

It's an error if:

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

#### 7.3.14 Dynamic Cutter Compensation (G41.1 and G42.1)

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

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

- D~ is the tip radius multiplied by two

G41.1 and G42.1 function the same as G41 and G42, with the added scope of being able to ignore the tool table and to program the tool diameter.

#### Troubleshooting

It's an error if:

- Cutter Compensation is commanded to turn on when it is already on

### 7.3.15 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

### 7.3.16 Engrave Sequential Serial Number (G47)

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

- Z~ is the depth of cut of the engraving
- R~ is the retract height between character segments in the numbers
- X~ is, if present, the starting X position, or the left side of the serial number  
If omitted, the current X position is assumed.
- Y~ is, if present, the starting Y position, or the bottom side of the serial number  
If omitted, the current Y position is assumed.
- P~ is, if present, the X extent (width) in current units (inches or millimeters) of the engraved number
- Q~ is, if present, the Y extent (height) in current units (inches or millimeters) of the engraved number
- D~ is, if present, the requested number of decimals of the engraved number

If the requested D value exceeds the number of decimals in the serial number, the serial number will show leading zeros. If the requested D value is less than the number of decimals in the serial number, only the digits of the serial number will show.



#### EXAMPLE

A serial number of 10, where D = 4, engraves as 0010. A serial number of 9056, where D = 3, engraves as 9056.

#### Troubleshooting

It's an error if:

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

### 7.3.17 Cancel Tool Length Compensation (G49)

To cancel tool length compensation, program: G49

### 7.3.18 Absolute Coordinates (G53)

For rapid linear motion to a point expressed in absolute coordinates, program: G01 G53 X~ Y~ Z~ (or use with G00 instead of G01)

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

#### Troubleshooting

It's an error if:

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

# 7: PROGRAMMING

## 7.3 Programming G-Code

### 7.3.19 Select Work Offset Coordinate System (G54 to G54.1 P500)

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

- To select work offset 1, program: G54 or G54.1 P1
- To select work offset 2, program: G55 or G54.1 P2
- To select work offset 3, program: G56 or G54.1 P3
- To select work offset 4, program: G57 or G54.1 P4
- To select work offset 5, program: G58 or G54.1 P5
- To select work offset 6, program: G59 or G54.1 P6
- To select work offset 7, program: G59.1 or G54.1 P7
- To select work offset 8, program: G59.2 or G54.1 P8
- To select work offset 9, program: G59.3 or G54.1 P9
- To select a work offset beyond the standard 9 (listed above), program: G54.1 P###, where P### is a parameter indicating the index of the work offset you want to use (work offset 10 through work offset 500).



#### EXAMPLE

To select the 124th work offset, program G54.1 P124.

For information, see "About Work Offsets" (page 118).

### 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).

### 7.3.20 Set Exact Path Control Mode (G61)

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

### 7.3.21 Set Blended Path Control Mode (G64)

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

- P~ is, if present, the maximum acceptable tool path deviation to round corners to maintain speed. If P is omitted then the speed is maintained however far from the programmed path the tool cuts.

- Q~ is, if present, the maximum deviation from collinearity that will collapse a series of linear G01 moves at the same feed rate into a single linear move.

It's okay to program for the mode that is already active.

### 7.3.22 Distance Mode (G90 and G91)

Interpretation of the operating system code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program: G90.

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

To go into incremental distance mode, program: G91.

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

### 7.3.23 Arc Distance Mode (G90.1 and G91.1)

G90.1 – Absolute distance mode for I and K offsets. When G90.1 is in effect, I and K both must be specified with G02/G03 for the XZ plane or it is an error.

G91.1 – Incremental distance mode for I and K offsets.

G91.1 returns I and K to their default behavior.

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



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

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

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

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

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

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

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



### EXAMPLE

If the current controlled point is at  $X = 4$ , and there is currently no G92 offset active, and then G92 X7 is programmed, this reassigns the current controlled point to  $X = 7$  — effectively moving the origin of the active coordinate system -3 units in X. The origins of all inactive coordinate systems also move -3 units in X. This -3 is saved in parameter 5211.

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

It's an error if:

- All axis words are omitted

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

To reset axis offsets to zero and sets parameters 5211 - 5219 to zero, program: G92.1

To reset axis offsets to zero, program: G92.2

To set the axis offset to the values saved in parameters 5211 to 5219, program: G92.3

### 7.3.25 Feed Rate Mode (G93, G94, and G95)

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

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

### 7.3.26 Spindle Control Mode (G96 and G97)

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

- D~ is the maximum spindle RPM.  
This word is optional.
- S~ is the surface speed.



**Note:** If G20 is the active mode, the value is interpreted as feet per minute. If G21 is the active mode, the value is interpreted as meters per minute



### EXAMPLE

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

# 7: PROGRAMMING

## 7.3 Programming G-Code

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

### 7.4 PROGRAMMING CANNED CYCLES

Read the following sections for reference:

|                                                |            |
|------------------------------------------------|------------|
| <b>7.4.1 Canned Cycles Reference</b> .....     | <b>141</b> |
| <b>7.4.2 High Speed Peck Drill (G73)</b> ..... | <b>142</b> |
| <b>7.4.3 Cancel Canned Cycles (G80)</b> .....  | <b>142</b> |
| <b>7.4.4 Drilling Cycle (G81)</b> .....        | <b>142</b> |
| <b>7.4.5 Simple Drilling Cycle (G82)</b> ..... | <b>144</b> |
| <b>7.4.6 Peck Drilling Cycle (G83)</b> .....   | <b>144</b> |
| <b>7.4.7 Boring Cycle (G85)</b> .....          | <b>144</b> |
| <b>7.4.8 Boring Cycle (G86)</b> .....          | <b>144</b> |
| <b>7.4.9 Boring Cycle (G88)</b> .....          | <b>144</b> |
| <b>7.4.10 Boring Cycle (G89)</b> .....         | <b>144</b> |

#### 7.4.1 Canned Cycles Reference

##### Supported Canned Cycles

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

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

- $\tilde{X}$  is the X-axis coordinate
- $\tilde{Y}$  is the Y-axis coordinate
- $\tilde{Z}$  is the Z-axis coordinate
- $\tilde{A}$  is the A-axis coordinate
- $\tilde{R}$  is the retract position along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane)
- $\tilde{L}$  is the L number is optional and represents the number of repeats

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

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

The R number is always sticky — it keeps its value on subsequent blocks if they're not explicitly programmed to be different.

In absolute distance mode (G90), the X, Y, R, and Z numbers are absolute positions in the current coordinate system.

In incremental distance mode (G91), when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ- or XZ-plane is selected, treatment of the axis words is analogous. Many canned cycles use the L word. The L word is optional and represents the number of repeats. L0 is not allowed. The L word is not sticky. The interpretation of the L word depends on the active distance mode:

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

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

##### Troubleshooting

It's an error if:

- X, Y, and Z words are all missing during a canned cycle
- A P number is required and a negative P number is used

## 7: PROGRAMMING

### 7.4 Programming Canned Cycles

- An L number is used that does not evaluate to a positive integer
- Rotational axis motion is used during a canned cycle
- Inverse time feed rate is active during a canned cycle
- Cutter radius compensation is active during a canned cycle

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

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

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

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

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

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

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

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

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

#### 7.4.2 High Speed Peck Drill (G73)

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

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

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

The G73 cycle is as follows:

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

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

#### Troubleshooting

It's an error if:

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

#### 7.4.3 Cancel Canned Cycles (G80)

To cancel all canned cycles, program: G80

It's okay to program G80 if no canned cycles are in effect.

After a G80, the motion mode must be set with G00 (or any other motion mode G word). If motion mode is not set after G80, this error message appears: **"Cannot use axis values without a g code that uses them."**

#### 7.4.4 Drilling Cycle (G81)

The G81 cycle is intended for drilling.

Program: G81 X~ Y~ Z~ A~ R~ L~

The G81 Cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Retract the Z-axis at traverse rate to clear Z.

#### Examples

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



### EXAMPLE

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

```
G90 G81 G98 X4 Y5 Z1.5 R2.8
```

This means that it's in absolute distance mode (G90), old Z retract mode (G98) and the G81 drilling cycle is performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. The following moves take place:

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



### EXAMPLE

The current position is (1, 2, 3), the XY-plane has been selected, the following line of code is interpreted: `G91`

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

This means that it's in incremental distance mode (G91), old Z retract mode, and the G81 drilling cycle is repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the initial Y position is 7 (=2+5), the clear Z position is 4.8 (=1.8+3) and the Z position is 4.2 (=4.8-0.6). Old Z is 3.0.

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

The first repeat consists of three moves:

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

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

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

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

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

Example code using G81 cycle:

```
(Sample Program G81EX18:)  
(Workpiece Size: X4, Y3, Z1)  
(Tool: Tool #6, 3/4" HSS DRILL)  
(Tool Start Position: X0, Y0, Z1)  
N2 G90 G80 G40 G54 G20 G17 G94 G64 (Safety Block)  
N5 G90 G80 G20  
N10 M06 T6 G43 H6  
N15 M03 S1300  
N20 G00 X1 Y1  
N25 Z0.5  
N30 G81 Z-0.25 R0.125 F5 (Drill Cycle Invoked)  
N35 X2  
N40 X3  
N45 Y2
```

# 7: PROGRAMMING

## 7.4 Programming Canned Cycles

```
N50 X2
N55 X1
N60 G80 G00 Z1 (Cancel Canned Cycles)
N65 X0 Y0
N70 M05
N75 M30
```

### 7.4.5 Simple Drilling Cycle (G82)

The G82 cycle is intended for drilling.

Program: G82 X~ Y~ Z~ A~ R~ L~ P~

The G82 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Dwell for the P number of seconds.
- × **Step 4:** Dwell for the P number of seconds.

### 7.4.6 Peck Drilling Cycle (G83)

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum).

Program: G83 X~ Y~ Z~ A~ R~ L~ Q~

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

The G83 cycle is as follows:

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

#### Troubleshooting

It's an error if:

- The Q number is negative or zero

### 7.4.7 Boring Cycle (G85)

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling.

Program: G85 X~ Y~ Z~ A~ R~ L~

The G85 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Retract the Z-axis at the current feed rate to clear Z.

### 7.4.8 Boring Cycle (G86)

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

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

The G86 cycle is as follows:

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

#### Troubleshooting

It's an error if:

- The spindle is not turning before this cycle is executed

### 7.4.9 Boring Cycle (G88)

The G88 cycle is intended for boring and uses a P word, where P specifies the number of seconds to dwell.

Program: G88 X~ Y~ Z~ A~ R~ L~ P~

The G88 cycle is as follows:

- × **Step 1:** Preliminary canned cycle motion.
- × **Step 2:** Move the Z-axis only at the current feed rate to the Z position.
- × **Step 3:** Dwell for the P number of seconds.
- × **Step 4:** Stop the spindle turning.
- × **Step 5:** Stop the program so the operator can retract the spindle manually.
- × **Step 6:** Restart the spindle in the direction it was going.

### 7.4.10 Boring Cycle (G89)

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell.

Program: G89 X~ Y~ Z~ A~ R~ L~ P~

The G89 cycle is as follows:

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

# 7: PROGRAMMING

## 7.5 Programming M-Code

### 7.5 PROGRAMMING M-CODE

Read the following sections for reference:

|                                                                   |     |
|-------------------------------------------------------------------|-----|
| 7.5.1 Supported M-Codes Reference .....                           | 146 |
| 7.5.2 Program Stop and Program End (M00, M01, M02, and M30) ..... | 146 |
| 7.5.3 Spindle Control (M03 and M05) .....                         | 147 |
| 7.5.4 Tool Change (M06) .....                                     | 147 |
| 7.5.5 Coolant Control (M08 and M09) .....                         | 148 |
| 7.5.6 Override Control (M48 and M49) .....                        | 148 |
| 7.5.7 Feed Override Control (M50) .....                           | 148 |
| 7.5.8 Spindle Speed Override Control (M51) .....                  | 148 |
| 7.5.9 Set Current Tool Number (M61) .....                         | 148 |
| 7.5.10 Set Output State (M64 and M65) .....                       | 148 |
| 7.5.11 Wait on Input (M66) .....                                  | 148 |
| 7.5.12 USB Camera Control (M301, M302, M303) .....                | 149 |

#### 7.5.1 Supported M-Codes Reference

| M-Code                                                                                                                  | Description                     |
|-------------------------------------------------------------------------------------------------------------------------|---------------------------------|
| <u>M00</u>                                                                                                              | Program stop                    |
| <u>M01</u>                                                                                                              | Optional program stop           |
| <u>M02</u>                                                                                                              | Program end                     |
| <u>M03</u>                                                                                                              | Rotate spindle clockwise        |
| <u>M05</u>                                                                                                              | Stop spindle rotation           |
| <u>M07</u> ,<br><u>M08</u>                                                                                              | Coolant on                      |
| <u>M09</u>                                                                                                              | All coolant off                 |
| <u>M30</u>                                                                                                              | Program end and rewind          |
| <u>M48</u>                                                                                                              | Enable speed and feed override  |
| <u>M49</u>                                                                                                              | Disable speed and feed override |
| <u>M64</u>                                                                                                              | Activate output relays          |
| <u>M65</u>                                                                                                              | Deactivate output relays        |
| <u>M66</u>                                                                                                              | Wait on an input                |
| <p><b>Note:</b> <u>M64</u> through <u>M66</u> is only useful with a <u>USB M-Code I/O Interface Kit (PN 32616)</u>.</p> |                                 |

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

#### 7.5.2 Program Stop and Program End (M00, M01, M02, and M30)

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

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

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

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

To end a program, program: M02 or M30.

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

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

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

### Display Information and Capture Images During an M00 or M01 Break

#### Display Information with Images

If the comment occurs on a line with **M00** or **M01**, and contains a file name with a .jpg or .png extension, PathPilot displays the 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 78).

#### 7.5.3 Spindle Control (M03 and M05)

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

The speed is programmed by the S word.

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

It's okay to use **M03** if the spindle speed is set to 0; if this is done, the spindle won't start turning. If later the spindle speed is set above 0, the spindle starts turning. It is permitted to use **M03** when the spindle is already turning, or to use **M05** when the spindle is already stopped.

#### 7.5.4 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.

# 7: PROGRAMMING

## 7.5 Programming M-Code

We recommend putting the `T~`, the `M06`, and the `G43 H~` on one line (block) of code.

 **EXAMPLE**  
`N191 M06 T3 G43 H3`

### 7.5.5 Coolant Control (M08 and M09)

To turn flood coolant on, program: `M08`

To turn all coolant off, program: `M09`

It's always okay to use any of these commands, regardless of what coolant is on or off.

### 7.5.6 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.

### 7.5.7 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).

### 7.5.8 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.

### 7.5.9 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

### 7.5.10 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
```

### 7.5.11 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.

#### 7.5.12 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 78).

# 7: PROGRAMMING

## 7.6 Programming Input Codes

### 7.6 PROGRAMMING INPUT CODES

Read the following sections for reference:

|                                    |     |
|------------------------------------|-----|
| 7.6.1 Feed Rate (F) .....          | 150 |
| 7.6.2 Spindle Speed (S) .....      | 150 |
| 7.6.3 Change Tool Number (T) ..... | 150 |

#### 7.6.1 Feed Rate (F)

To set the feed rate, program: `F~`  
Depending on the setting of the feed mode toggle, the rate may be in units-per-minute or units-per-rev of the spindle. The units are those defined by the G20/G21 mode. The feed rate may sometimes be overridden.

#### 7.6.2 Spindle Speed (S)

To set the speed in revolutions per minute (rpm) of the spindle, program: `S~`  
The spindle turns at the commanded speed when it has been programmed to start turning. It'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

#### 7.6.3 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

### 7.7 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:

|                                |            |
|--------------------------------|------------|
| <b>7.7.1 Parameters</b> .....  | <b>151</b> |
| <b>7.7.2 Expressions</b> ..... | <b>153</b> |
| <b>7.7.3 Subroutines</b> ..... | <b>154</b> |

#### 7.7.1 Parameters

Read the following sections for reference:

|                                              |            |
|----------------------------------------------|------------|
| <b>Parameters Reference</b> .....            | <b>151</b> |
| <b>Numbered Parameters Reference</b> .....   | <b>152</b> |
| <b>Subroutine Parameters Reference</b> ..... | <b>152</b> |
| <b>Named Parameters Reference</b> .....      | <b>152</b> |

#### 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

# 7: PROGRAMMING

## 7.7 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.

### 7.7.2 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..... | 153 |
| Functions Reference.....        | 153 |

#### 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)

# 7: PROGRAMMING

## 7.7 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

### 7.7.3 Subroutines

Subroutines are subprograms that are called from inside another program.

Read the following sections for reference:

|                                                |            |
|------------------------------------------------|------------|
| <b>Subroutines Reference</b> .....             | <b>154</b> |
| <b>Conditional Subroutines Reference</b> ..... | <b>155</b> |
| <b>Repeating Subroutines Reference</b> .....   | <b>155</b> |
| <b>Looping Subroutines Reference</b> .....     | <b>155</b> |

#### 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.

# 7: PROGRAMMING

## 7.7 Advanced Programming

### do/while

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

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

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

### while/endwhile

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

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

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

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

# MACHINE MAINTENANCE

## IN THIS SECTION, YOU'LL LEARN:

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



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

---

## CONTENTS

|                                 |     |
|---------------------------------|-----|
| 8.1 Maintenance Safety .....    | 158 |
| 8.2 Maintenance Schedules ..... | 159 |



**TORMACH®**

## 8: MACHINE MAINTENANCE

### 8.1 Maintenance Safety

#### 8.1 MAINTENANCE SAFETY

Read and understand the following safety messages before beginning any maintenance procedures.

##### 8.1.1 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+.

##### 8.1.2 Swarf Maintenance Procedures

- ✓ Wear work gloves.

## 8.2 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 161). For any additional support, we can help. Create a support ticket with Tormach Technical Support at [tormach.com/how-to-submit-a-support-ticket](https://tormach.com/how-to-submit-a-support-ticket) for guidance on how to proceed.

### 8.2.1 Daily

- ☐ Vacuum chips and dust from the gantry, bed, rails, and ballscrews to prevent buildup and wear.
- ☐ Wipe exposed linear rails and ballscrews with a dry microfiber cloth. Do not heavily oil daily.
- ☐ Inspect tool holders and ER collets for chip buildup. Remove any chips or debris inside the taper. Contamination can affect runout accuracy.
- ☐ Inspect the ATC tool forks and replace any that are cracked, bent, or otherwise damaged.
- ☐ Inspect cutter condition. Replace chipped tools to maintain cut quality and reduce spindle load.
- ☐ Empty the vacuum and dust separator as needed.

### 8.2.2 Monthly (or every ~80 hours)

- ☐ Check for software updates in PathPilot. From the **Status** tab, select **Check for Updates** to make sure the software is current.
- ☐ Apply light oil to the linear rails and blocks. Add a single drop of SAE 10W/ISO 32 way oil (or similar).
- ☐ Inspect the pneumatic system for leaks. Reseal any loose fittings with two wraps of PTFE tape (or similar) and tighten snugly plus a quarter-turn.
- ☐ Check the ATC oil reservoir. Fill the reservoir halfway with air tool oil (SAE 10W/ISO 32 or similar). Completely close the reservoir, then loosen a quarter-turn to set the appropriate fluid metering.

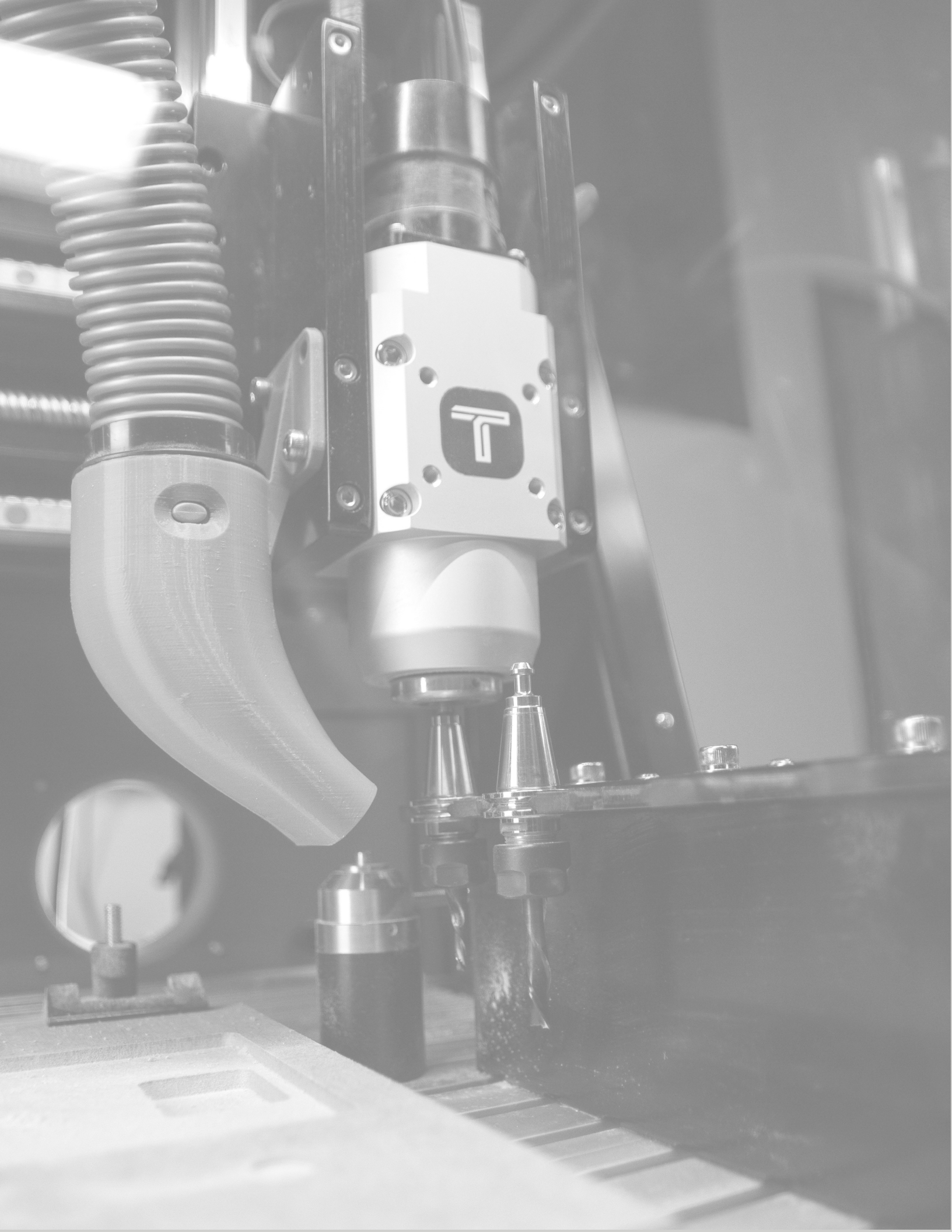
- ☐ Verify that the pneumatic inlet pressure is set to 90 psi (0.62 MPa).
- ☐ Verify that the MQL pressure (if equipped) is set to 15 psi (0.1 MPa).

### 8.2.3 Semi-Annually (or Every ~250 hours)

- ☐ Grease the ballscrews and linear guides with the provided grease gun and grease.
- ☐ Clean the cabinet filters to ensure proper airflow. MDF dust can clog the filters quickly.

### 8.2.4 Annually (or as needed)

- ☐ Replace the ER collets. We recommend replacing these every year or after 500 tool replacements (whichever comes first).
- ☐ Examine the windows and replace any that are cracked or otherwise damaged.
- ☐ Examine the ISO 10 tool holders and replace any that are damaged.
- ☐ Check the spindle runout using a tenths indicator. The total indicated runout (TIR) shouldn't exceed 0.0008 in. at the collet face.
- ☐ Verify machine squareness and tram using a 1-2-3 block or a spindle tram indicator.
- ☐ Inspect the table fasteners and the frame squareness. Tighten any loose fasteners. Vibration during operation can cause hardware to loosen over time.



# 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

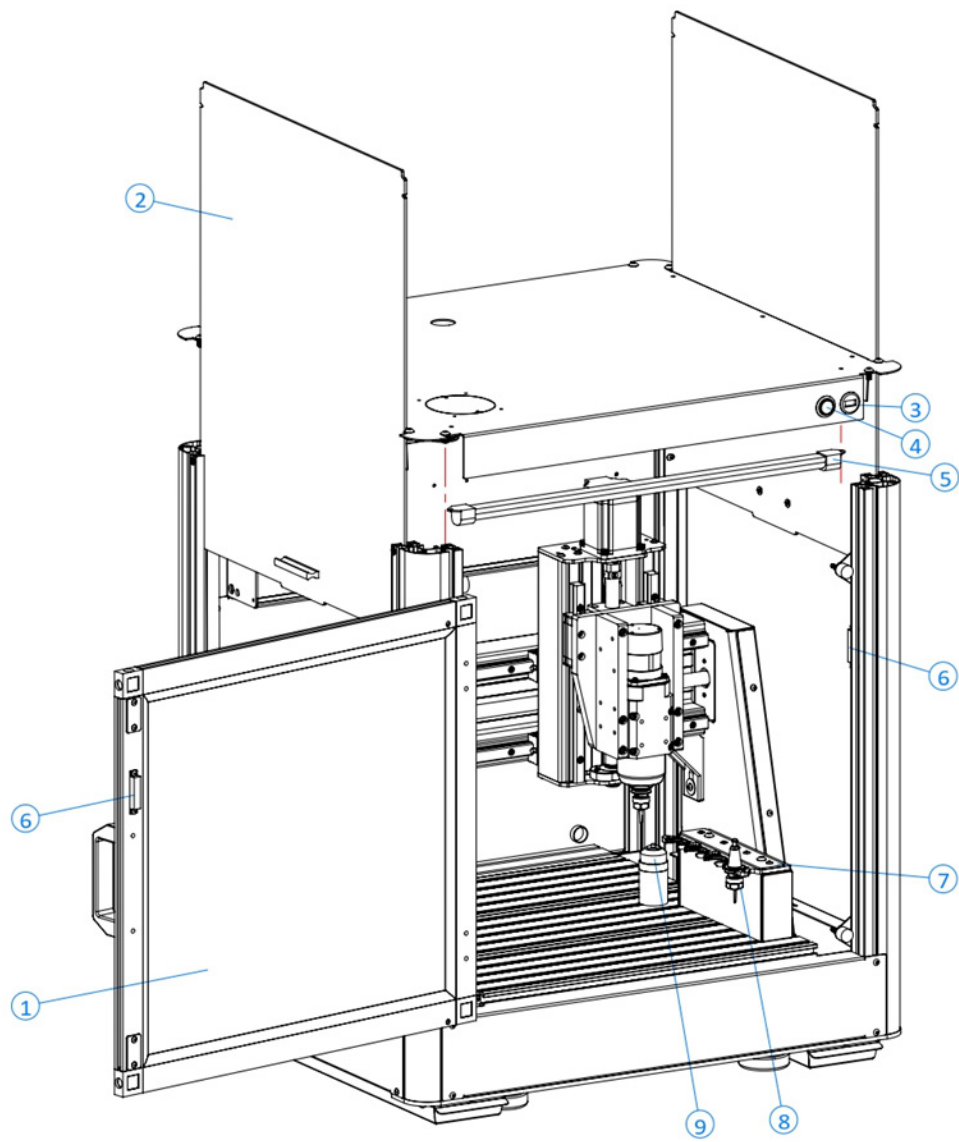
|                      |     |
|----------------------|-----|
| 9.1 xsTech Pro ..... | 162 |
|----------------------|-----|

9: DIAGRAMS AND PARTS LISTS

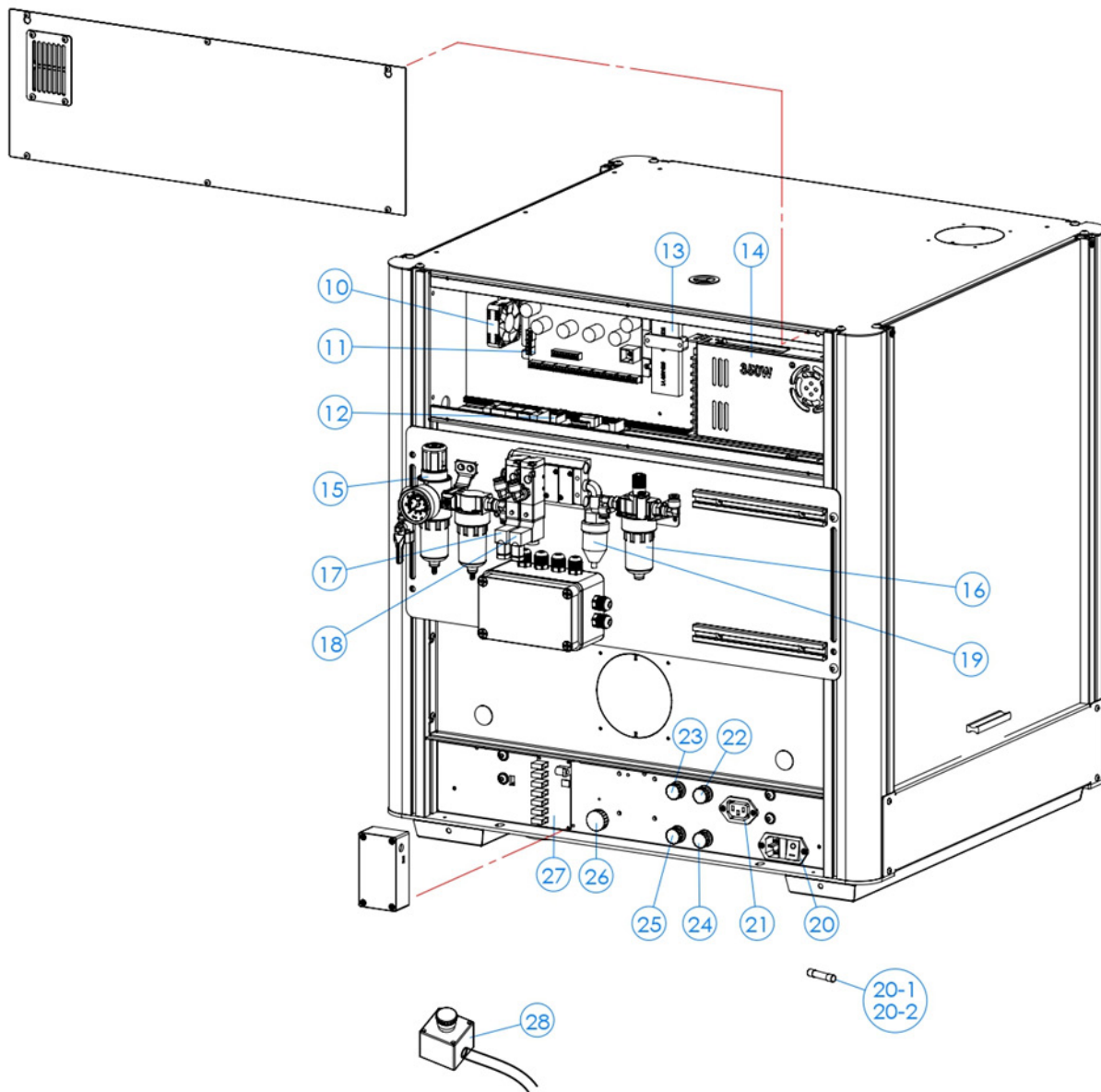
9.1 xsTech Pro

9.1 XSTECH PRO

9.1.1 Front View



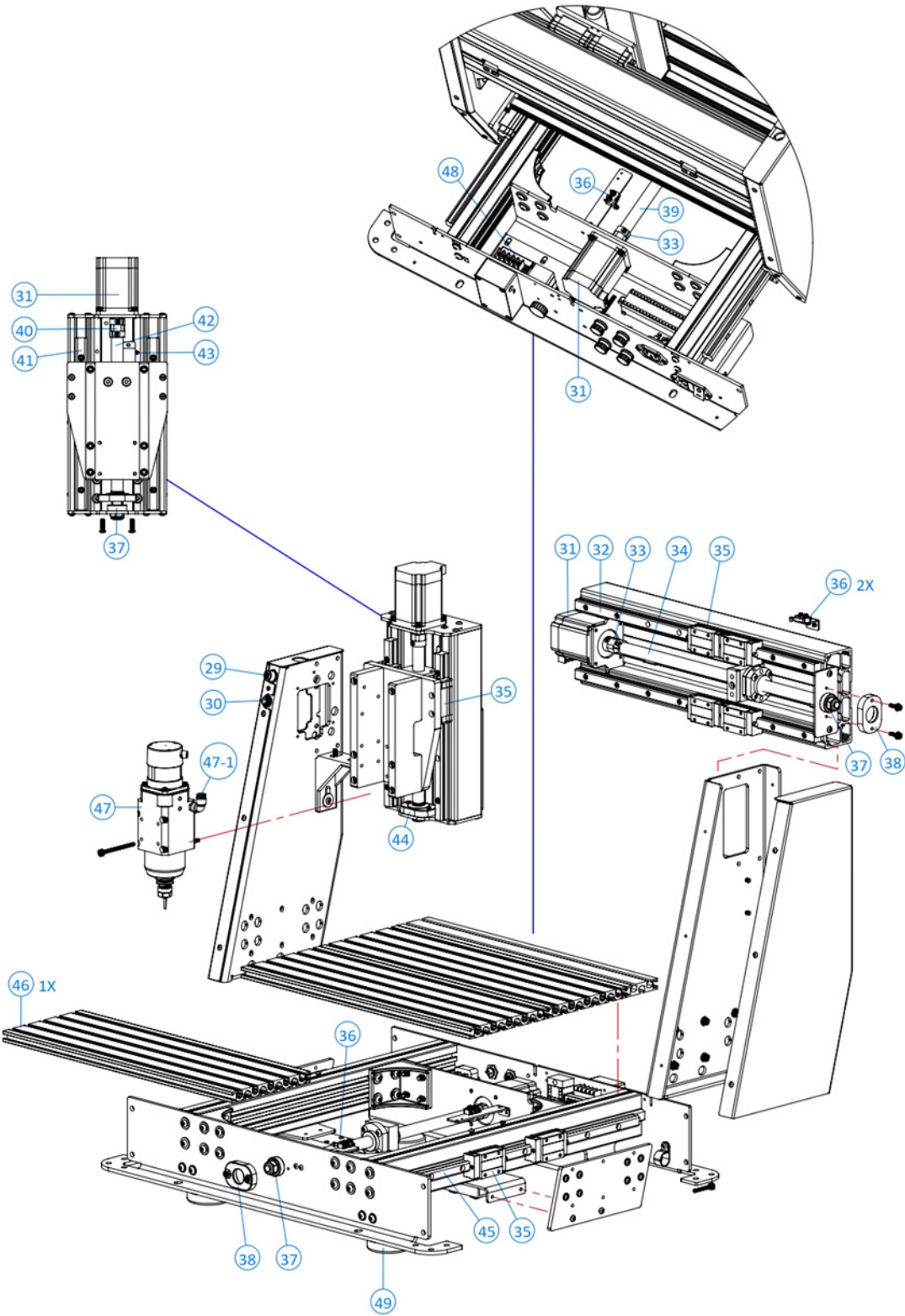
### 9.1.2 Rear View



9: DIAGRAMS AND PARTS LISTS

9.1 xsTech Pro

9.1.3 Machine View (Enclosure Removed)



| ID   | Description                                                                       | Quantity |
|------|-----------------------------------------------------------------------------------|----------|
| 1    | Front Window (PN 54797)                                                           | 1        |
| 2    | Side Window (PN 54798)                                                            | 1        |
| 3    | USB Port 2.0                                                                      | 1        |
| 4    | Power Drawbar Button (PN 54807)                                                   | 1        |
| 5    | LED Light (PN 55191)                                                              | 1        |
| 6    | Door Close Sensor (PN 54808)                                                      | 1        |
| 7    | Tool Fork (PN 54829)                                                              | 1        |
| 8    | ISO10 Tool Holder                                                                 | 1        |
| 9    | Tool Setter (PN 54799)                                                            | 1        |
| 10   | Fan (PN 55188)                                                                    | 1        |
| 11   | Torstep Board (PN 55186)                                                          | 1        |
| 12   | Brake Board (PN 55190)                                                            | 1        |
| 13   | BLDC Spindle Drive (PN 54835)                                                     | 1        |
| 14   | Power Supply 350 W 24 V (PN 54834)                                                | 1        |
| 15   | FRL Assembly (PN 54832)                                                           | 1        |
| 16   | Second FRL (PN 54833)                                                             | 1        |
| 17   | Solenoid (ATC) (PN 54828)                                                         | 1        |
| 18   | Solenoid (Mist)                                                                   | 1        |
| 19   | Pressure Sensor                                                                   | 1        |
| 20   | 3-in-1 Power Switch (AC Input Power, Fuse × 2 USA Spec., Power Switch) (PN 55184) | 1        |
| 20-1 | Fuse USA Spec.                                                                    | 1        |
| 20-2 | Fuse EU Spec.                                                                     | 1        |
| 21   | IEC Connector (Controller Power) (PN 55185)                                       | 1        |
| 22   | 2-Pin Connector for Emergency Stop (PN 54820)                                     | 1        |
| 23   | 7-Pin Connector for Pneumatic Module (PN 54825)                                   | 1        |
| 24   | 4-Pin Connector for AC Contactor Module (PN 54822)                                | 1        |
| 25   | 4-Pin Connector for 24 Vdc I/O Module                                             | 1        |
| 26   | Ethernet Bulkhead Connector (PN 54826)                                            | 1        |
| 27   | HUS HUB 7-Port USB 2.0 (PN 54827)                                                 | 1        |
| 28   | Emergency Stop Box (PN 54821)                                                     | 1        |
| 29   | Spindle Probe Connector (with Cap, Wire, Quick-Connector) (PN 54801)              | 1        |

## 9: DIAGRAMS AND PARTS LISTS

### 9.1 xsTech Pro

| ID   | Description                                                     | Quantity |
|------|-----------------------------------------------------------------|----------|
| 30   | 4th Axis Connector (with Cap, Wire, Quick-Connector) (PN 54800) | 1        |
| 31   | X-, Y-, Z-Axis Motor (PN 54802)                                 | 1        |
| 32   | X-Axis Linear Rails Set (PN 54816)                              | 1        |
| 33   | X, Y Ball Screw Coupler (PN 54809)                              | 1        |
| 34   | X-Axis Ball Screw (PN 54817)                                    | 1        |
| 35   | X-, Y-, Z-Axis Linear Bearing Blocks (PN 54831)                 | 1        |
| 36   | X-, Y-Axis Limit Switch Set (PN 54818)                          | 1        |
| 37   | Ball Screw Support Bearing (PN 54806)                           | 1        |
| 38   | X-, Y-Axis Ball Screw Support Bearing Housing                   | 1        |
| 39   | Y-Axis Ball Screw (PN 54837)                                    | 1        |
| 40   | Z Ball Screw Coupler (PN 54843)                                 | 1        |
| 41   | Z-Axis Linear Rails Set (PN 54813)                              | 1        |
| 42   | Z-Axis Ball Screw (PN 54814)                                    | 1        |
| 43   | Z-Axis Limit Switch (PN 54815)                                  | 1        |
| 44   | Z-Axis Ball Screw Support Bearing Housing (PN 54830)            | 1        |
| 45   | Y-Axis Linear Rails Set (PN 54836)                              | 1        |
| 46   | Table Extrusions (PN 55189)                                     | 1        |
| 47   | Spindle Assembly with BLDC Motor (PN 54810)                     | 1        |
| 47-1 | Power Drawbar Push-to-Connect Connector (PN 54812)              | 1        |
| 48   | 5 V Power Supply (PN 55187)                                     | 1        |
| 49   | Foot Set (PN 54839)                                             | 1        |

# ELECTRICAL SCHEMATICS

## IN THIS SECTION, YOU'LL LEARN:

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

---

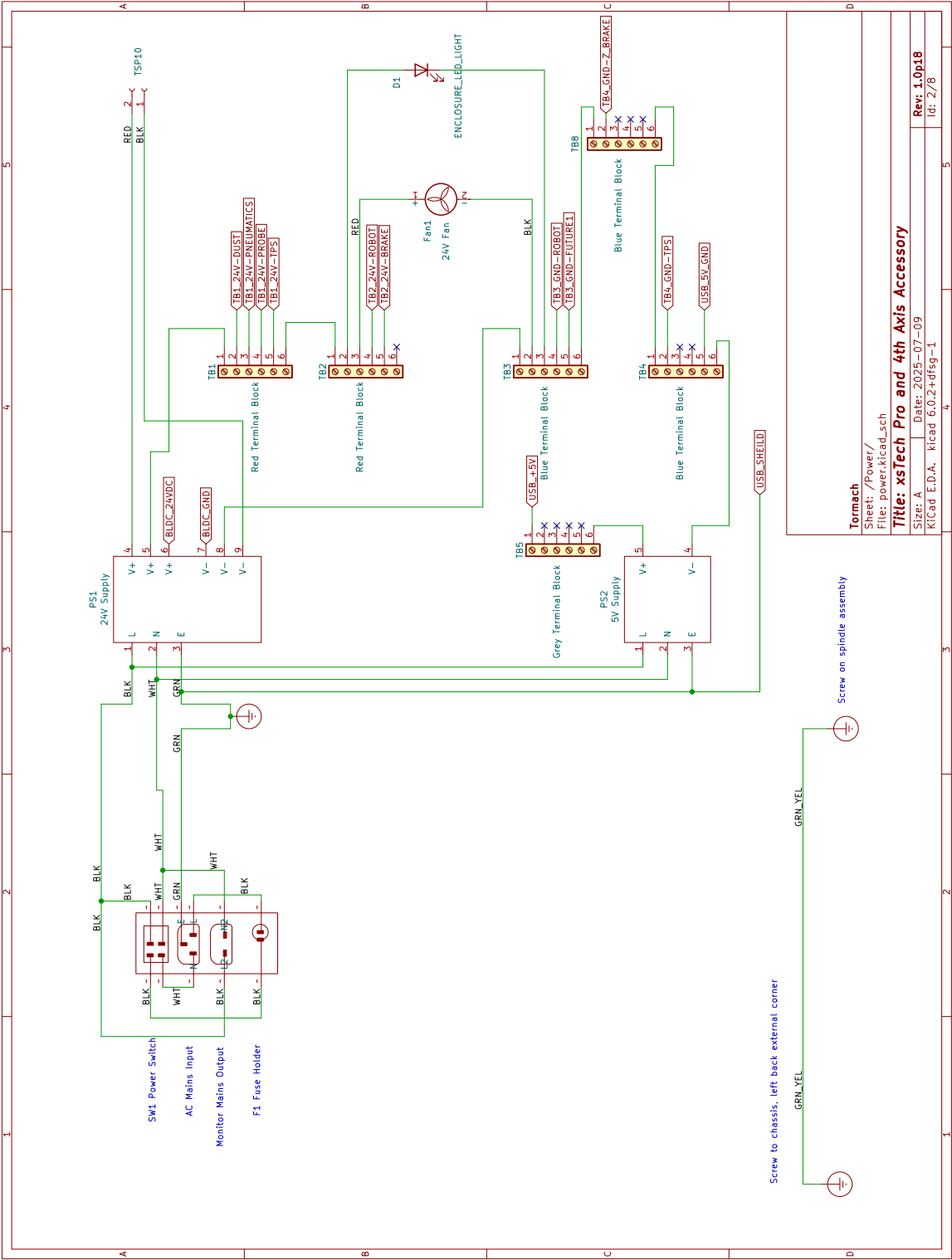
### CONTENTS

|                                          |     |
|------------------------------------------|-----|
| 10.1 Power (Page 2) .....                | 168 |
| 10.2 Switches (Page 3) .....             | 169 |
| 10.3 Spindle (Page 4) .....              | 170 |
| 10.4 Axis Motors (Page 5) .....          | 171 |
| 10.5 Ethernet Cable (Page 6) .....       | 172 |
| 10.6 Solenoids and Relays (Page 7) ..... | 173 |
| 10.7 USB Hub (Page 8) .....              | 174 |

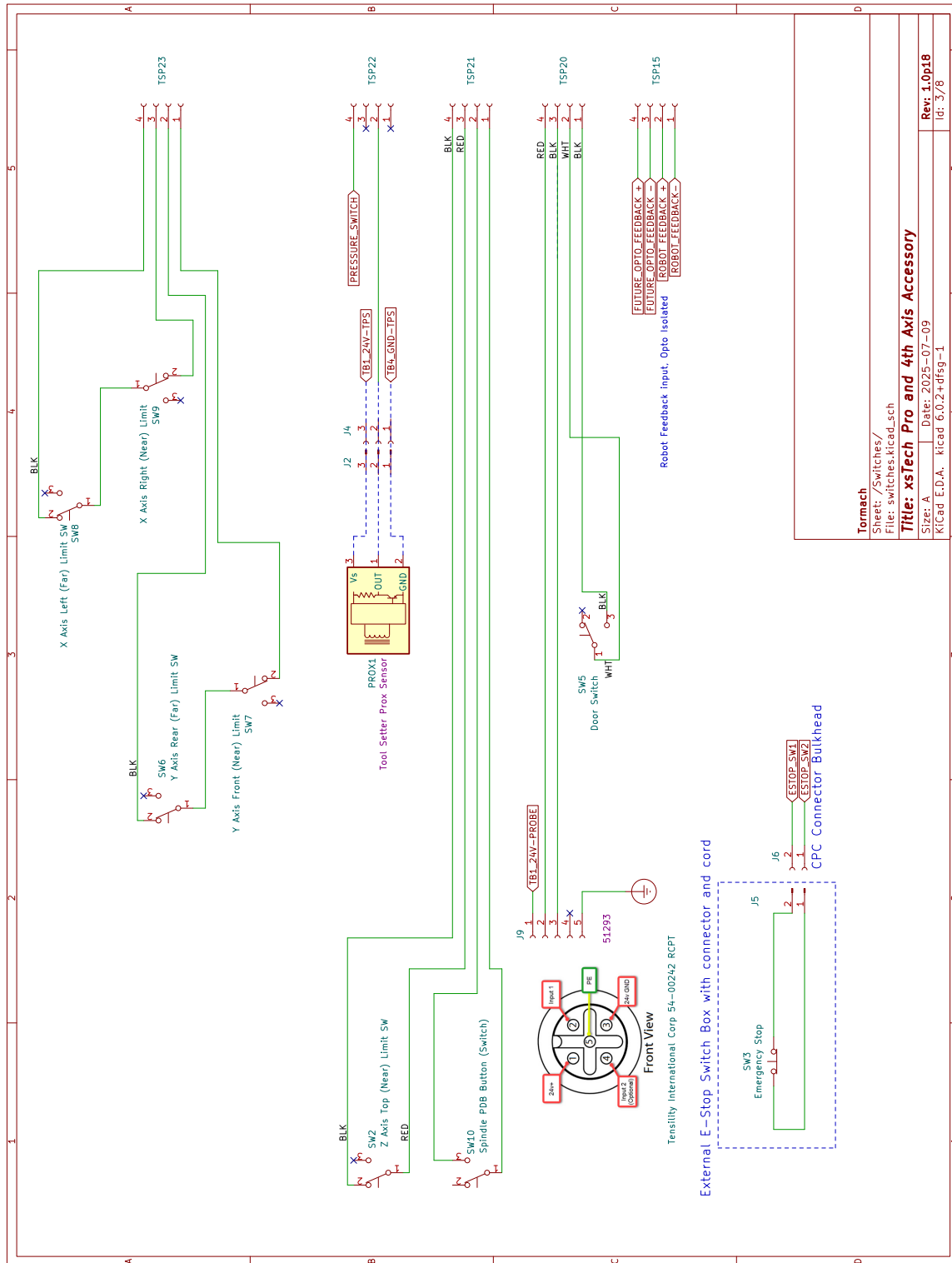
10: ELECTRICAL SCHEMATICS

10.1 Power (Page 2)

10.1 POWER (PAGE 2)



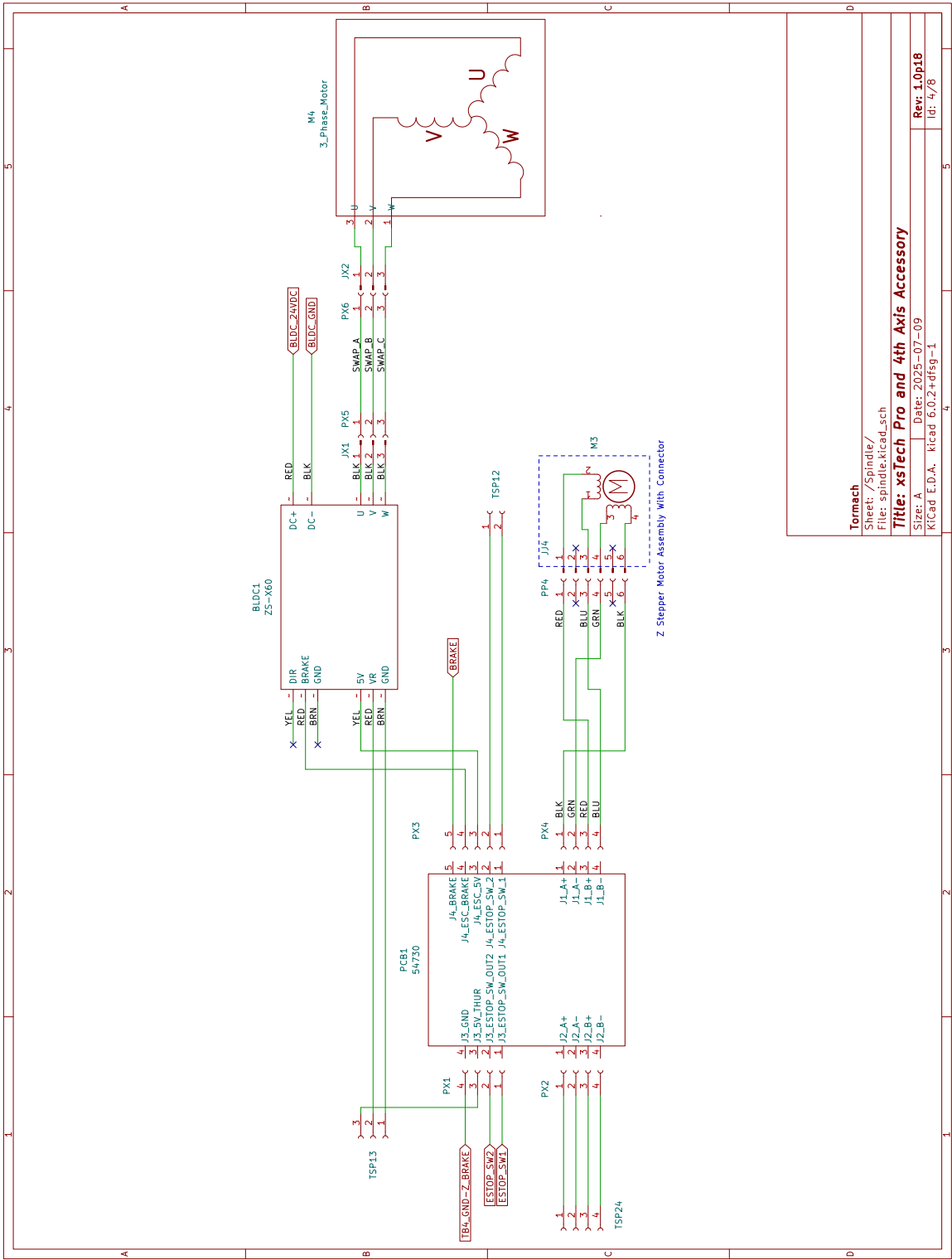
## 10: ELECTRICAL SCHEMATICS



10: ELECTRICAL SCHEMATICS

10.3 Spindle (Page 4)

10.3 SPINDLE (PAGE 4)

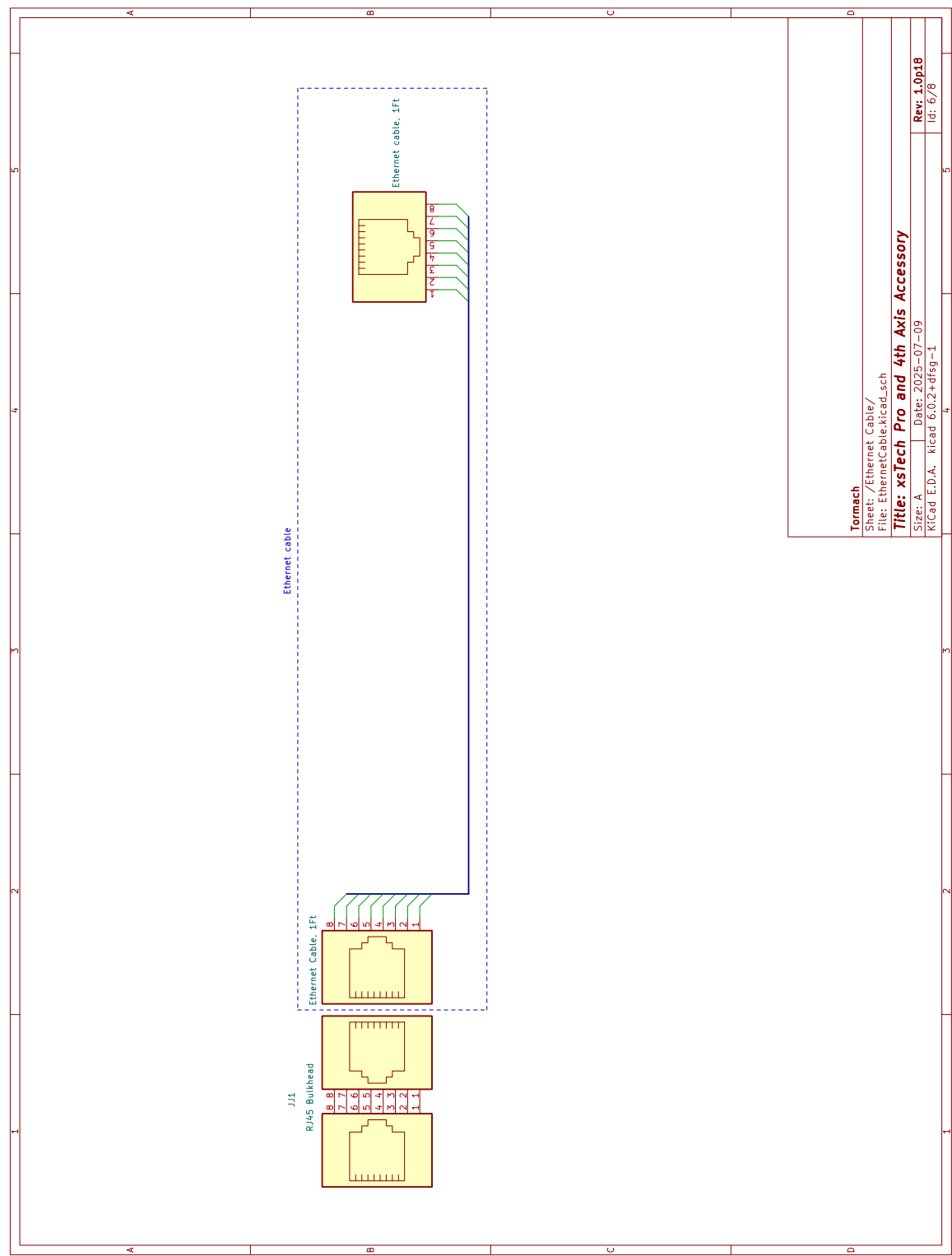




10: ELECTRICAL SCHEMATICS

10.5 Ethernet Cable (Page 6)

10.5 ETHERNET CABLE (PAGE 6)



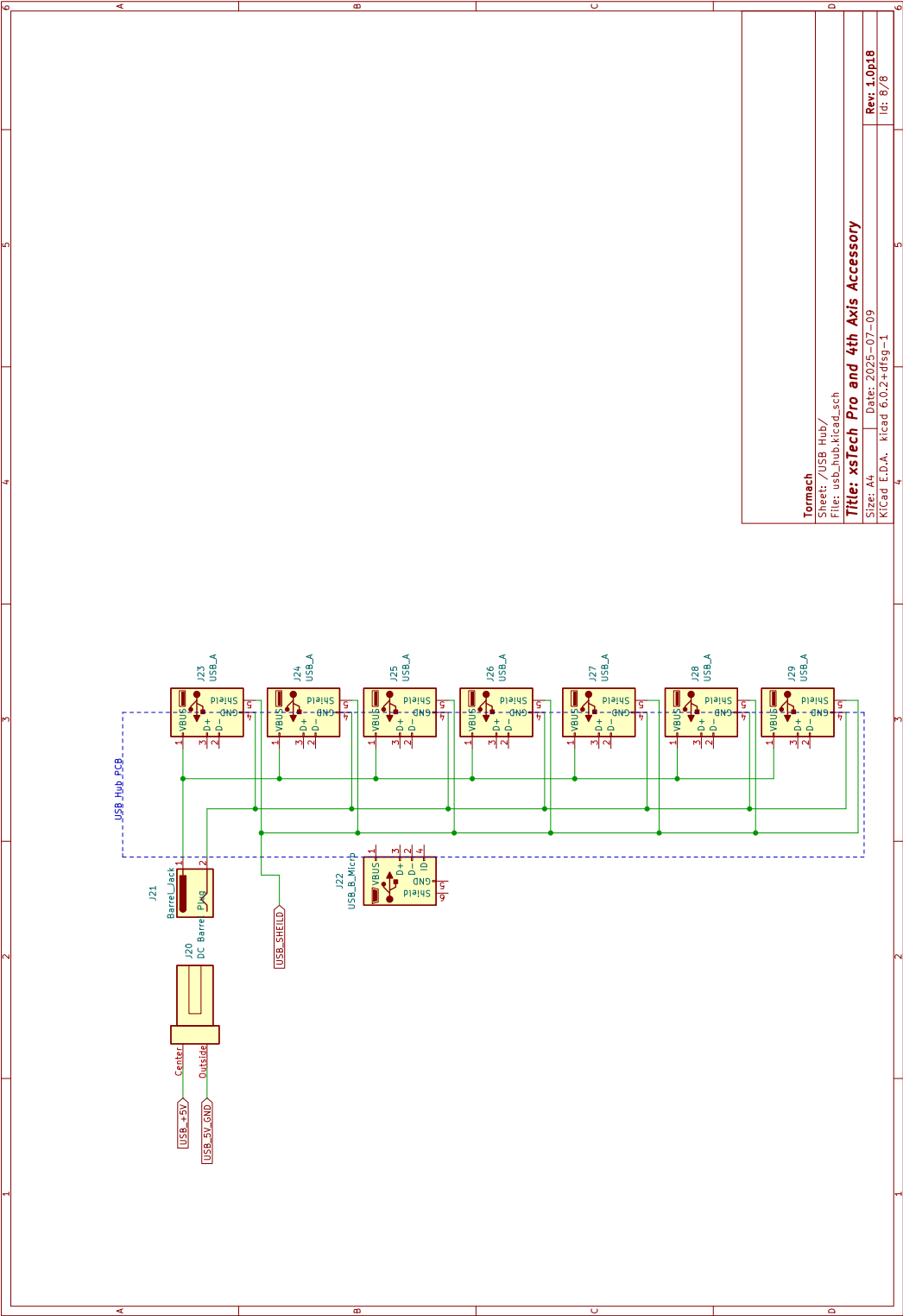
## 10.6 SOLENOIDS AND RELAYS (PAGE 7)



10: ELECTRICAL SCHEMATICS

10.7 USB Hub (Page 8)

10.7 USB HUB (PAGE 8)



|                                          |                    |             |
|------------------------------------------|--------------------|-------------|
| Tormach                                  |                    |             |
| Sheet: /USB Hub/                         |                    |             |
| File: usb_hub.kicad_sch                  |                    |             |
| Title: xsTech Pro and 4th Axis Accessory |                    |             |
| Size: A4                                 | Date: 2025-07-09   | Rev: 1.0n18 |
| Kicad E.D.A.                             | Kicad 6.0.2+dfsg-1 | Id: 8/8     |