PATHPILOT USER GUIDE FOR LATHE

RELEASE NOTES, INTERFACE OVERVIEW, TOOLS AND FEATURES, PROGRAMMING

TORMACH®

TORMACH.COM

Copyright

Notice

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

You're welcome to make copies of this document for evaluating, learning about, and/or using the machine. You may not charge for any copies you make beyond the cost of printing.

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

To the Reader

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

Credits

Tormach®, 15L Slant-PRO®, 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.

PREFACE

ABOUT THIS DOCUMENT

PURPOSE AND SCOPE

This document is intended to provide sufficient information to allow you to use your PathPilot controller. 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 at tormach.com/support.
3. If you still need answers, gather the following information so that we may help you as quickly as possible:
    - Your phone number, address, and company name (if applicable).
    - Machine model and serial number, which are located next to the Main Disconnect switch.
    - The version of PathPilot that you're running.
    - Any accessories that you have for your machine.
    - A clear and concise description of the issue.
    - Any supporting media and information that you can share with us. For example, you could:
        - Analyze what might have changed since the machine last worked correctly.
        - Record a short video.
        - Take a picture of a part.
        - For software, share log data .zip files, screen captures, or program files.
          For information, see "Share Log Data .zip Files" (below).
        - From the PathPilot interface, on the Status tab, record any available information.
        - Use a digital multimeter for voltage readings.
4. Once you've gathered the information in Step 3, contact us in the following ways:
    a. Create a support ticket: Go to tormach.com/how-to-submit-a-support-ticket
    b. Phone: (608) 849-8381 (Monday through Friday, 8 a.m. to 5 p.m. U.S. Central Standard Time)

SHARE LOG DATA .ZIP FILES

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

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

ADDITIONAL INFORMATION

For additional technical information and support videos, see tormach.com/support.

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.

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

TABLE OF CONTENTS

WHAT'S NEW?

IN THIS SECTION, YOU'LL LEARN:

- About the enhancements and fixed issues in the most recent version of PathPilot.

RELEASE NOTES FOR PATHPILOT V2.10.0

Fall 2023

Enhancements

All

We added:

- Support for a new on-screen keyboard. (PP-3497)
- Support for USB cameras.
  - "Dashcam" automatic 30 second video recording prior to hitting the emergency stop button. (PP-3525)
  - Manual video recording and image capture.
  - Video recording and image capture through the addition of three new M-codes.
  - A new M00/M01 workflow to simplify the creation of image files that can be shown during pause. (PP-3526)
- A new screen layout for portrait (vertical) orientation on 1920 × 1080 widescreen monitors. (PP-3522)
- Block delete functionality. You can now prefix any line of G-code with a slash (for example, `/G0 X0Y0Z0`). When you select the Block Delete button (toggle it on), PathPilot skips any line that begins with a slash (`/`). If Block Delete is toggled off, the lines execute as normal. (PP-3978)

Mills

- We split the **Coolant** button into a **Flood** and a **Mist** button on M, M+, and MX mills. For instructions on adding an additional mist outlet on your existing 1100/770 mill, refer to Tormach document SB10828. (PP-4139)

Fixed Issues

All

- We fixed an issue where saving files with a quotation mark (`"`) in the name prevented the file from loading. (PP-3701)

Mills

- We fixed an issue where incorrectly scaled feed rate values were shown in the DRO with G21 active during a pure A-axis motion. (PP-4115)

Lathes

- We fixed an issue where, in rare situations, the spindle on the 8L didn't stop when the enclosure door was opened. (PP-4091)

PATHPILOT INTERFACE OVERVIEW

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

TORMACH®

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.

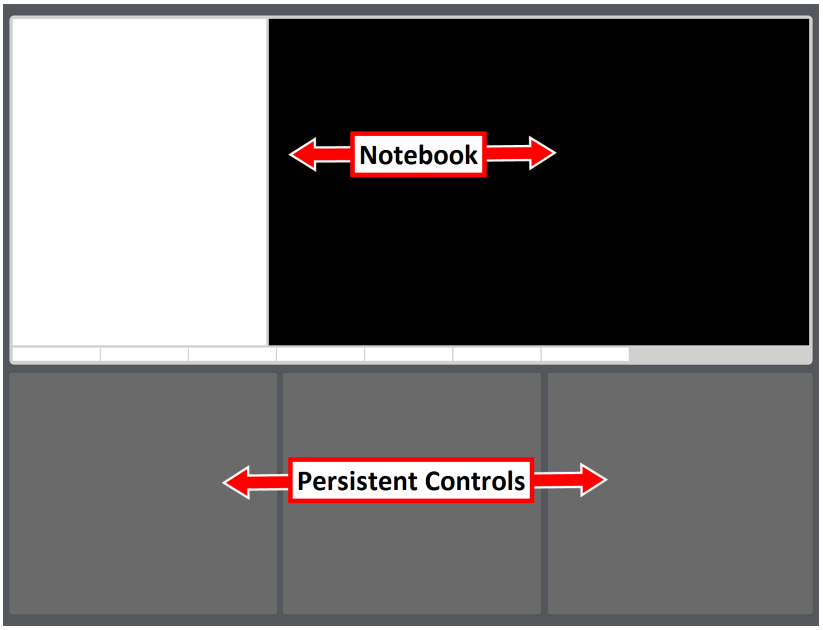


Figure 1: Sections in the PathPilot interface.

NOTEBOOK SECTION

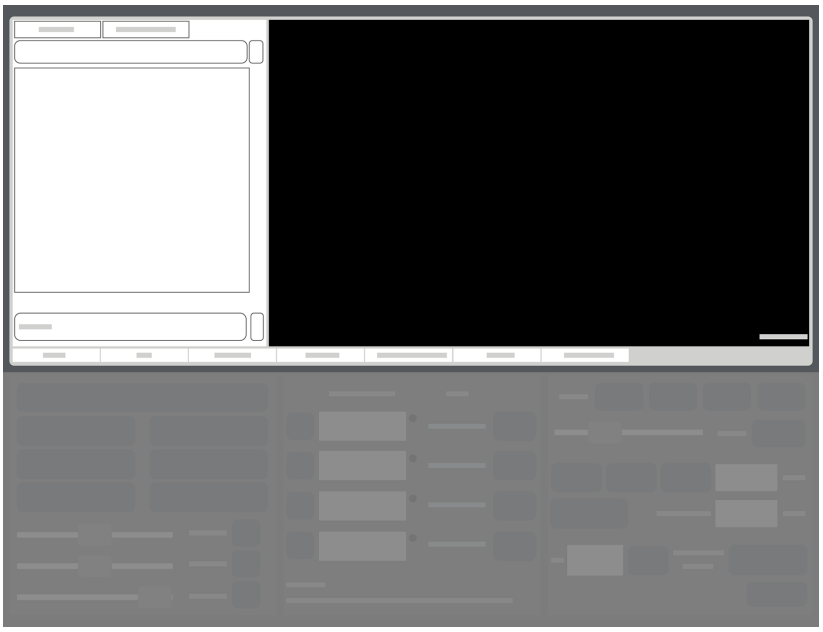


Figure 2: Notebook section.

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

Main Tab

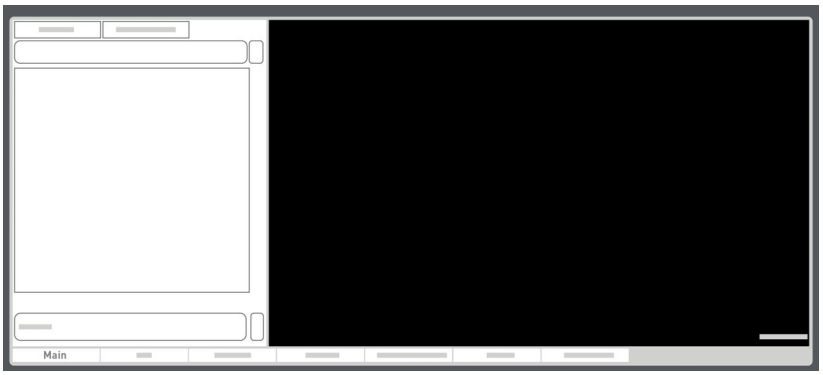


Figure 3: Main tab.

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 37); "Close the Current Program" (page 37).
- Send G-code commands directly to the machine using the Manual Data Input (MDI) Line DRO field.
  For information, see "Manually Enter Commands" (page 121).
- 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 42); "Expand the G-Code Tab" (page 41); "Change the View of the Tool Path Display" (page 44).

File Tab

Figure 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 35).
- Edit G-code files.
  For information, see "Edit G-Code" (page 37).
- Load .nc files into PathPilot to run a program.
  For information, see "Load G-Code" (page 35).
- Move files within the system.
  For information, see "Preview G-Code Files" (page 36); "Manage System Files" (page 130).

Settings Tab

Figure 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 78).
- Change the screen's layout orientation (landscape or portrait).
  For information, see "Change the Screen Orientation" (page 78).
- Configure PathPilot for the accessories you're using.
  For information, see "Enable the On-Screen Keyboard" (page 85); "Enable the USB M-Code I/O Interface Kit" (page 87); "Use a USB Camera" (page 88).
- Turn on feeds and speeds suggestions when using conversational programming.
  For information, see "Enable Feeds and Speeds Suggestions in Conversational Routines" (page 125).
- Disable the limit switches for troubleshooting.
  For information, see "Disable Limit Switches" (page 83).
- Specify the way in which you want to use a G30 move.
  For information, see "Limit G30 Moves" (page 84).
- Identify the available G-code modes that you can use.
  For information, see "View Available G-Code Modes" (page 98).

Offsets Tab

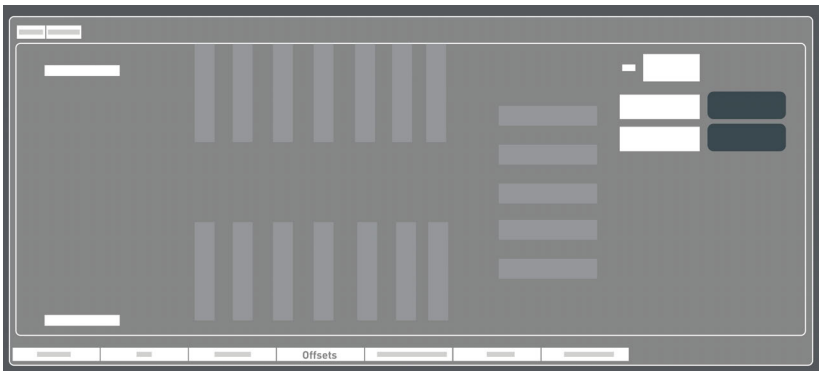


Figure 6: Offsets tab.

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

- Make and restore backup files of your settings.
  For information, see "Create Backup Files" (page 130); "Restore Backup Files" (page 133).
- Import and export .csv files of your tool table.
  For information, see "Import and Export the Tool Table" (page 134).
- Work with a table of tool descriptions and tool offsets.
  For information, see "Set Tool Geometry Offsets" (page 93).
- Read the currently programmed work offsets.
  For information, see "View Work Offsets" (page 98).
- Change the tool number.
  For information, see "Change the Tool Number" (page 118).

Conversational Tab

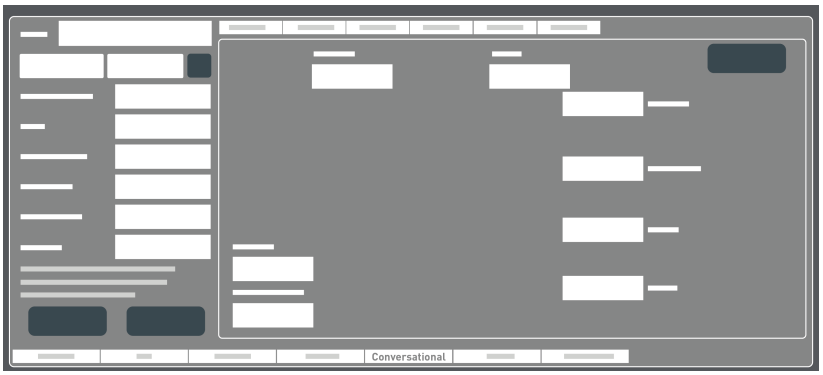


Figure 7: Conversational tab.

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 46).

Status Tab

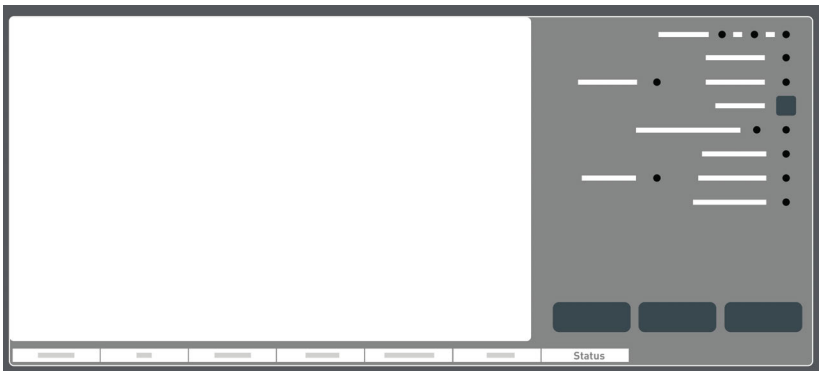


Figure 8: Status tab.

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

- View diagnostic machine information.
- Read error messages.
- Configure your internet connection.
  For information, see "Enable an Internet Connection" (page 74).
- Update or install a previous version of PathPilot.
  For information, see "Manage PathPilot Versions" (page 30).

PERSISTENT CONTROLS

Figure 9: Persistent Controls section.

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

Program Control Area

Figure 10: Program Control area.

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

- Reset the machine.
  For information, see "Bring the Machine Out of Reset" (page 99).
- Start, stop, or pause a G-code program.
  For information, see "Start a Program" (page 104); "Stop Machine Motion" (page 105); "Use the Feed Hold Function" (page 108).
- Use overrides to change the feed rate, spindle speed, and maximum velocity.
  For information, see "Use the Feed Rate Override Function" (page 109); "Use the Maxvel Override Function" (page 110); "Use the Spindle Override Function" (page 111).
- Manually control a G-code program.
  For information, see "Use M01 Break Mode" (page 110); "Use Single Block Mode" (page 111).

Position Status Area

Figure 11: Position Status area.

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

- Create work offsets.
  For information, see "Set Work Offsets" (page 96).
- Understand how you're jogging the machine.
  For information, see "View the Active Axis to Jog" (page 99); "View the Current Machine Position" (page 103); "View the Distance to Go" (page 107).
- Quickly determine which G-code modes are active.
  For information, see "View the Active G-Code Modes" (page 106).
- Change the spindle speed or feed rate.
  For information, see "Change the Feed Rate Mode" (page 115); "Change the Spindle Speed Command Mode" (page 116).

Manual Control Area

Figure 12: Manual Control area.

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

- Move the machine axes.
  For information, see "Jog the Machine" (page 100).
- Preset a G30 position.
  For information, see "Use a G30 Position" (page 119).
- Reference the machine axes.
  For information, see "Reference the Machine" (page 103).

KEYBOARD SHORTCUTS

The following table lists the keyboard shortcuts in PathPilot.

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

MANAGE PATHPILOT VERSIONS

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

- "Download and Install an Update File from the Controller" (below)
- "Install an Update File from a USB Drive" (on the next page)
- "Install a Previous Version of an Update File" (page 32)

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. Then, select **Update**.

Figure 13: Update button on the Status tab.

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

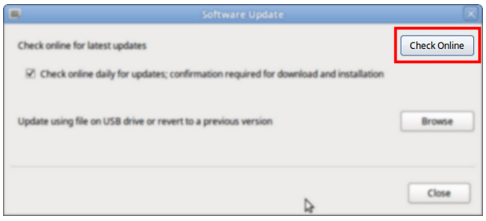


Figure 14: Software Update dialog box.

4. Select **Install**.

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

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

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

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 16: Update button on the Status tab.

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

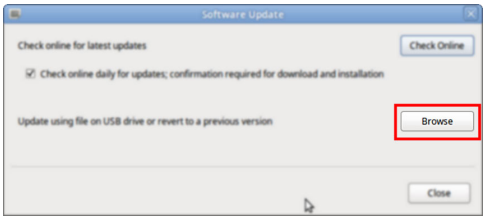


Figure 17: Software Update dialog box.

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

Figure 18: Browse dialog box.

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

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 19: Update button on the Status tab.

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

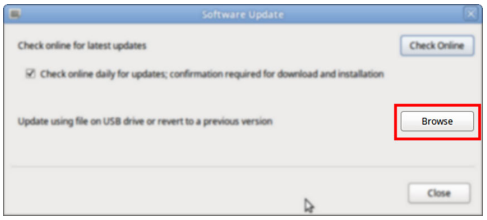


Figure 20: Software Update dialog box.

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

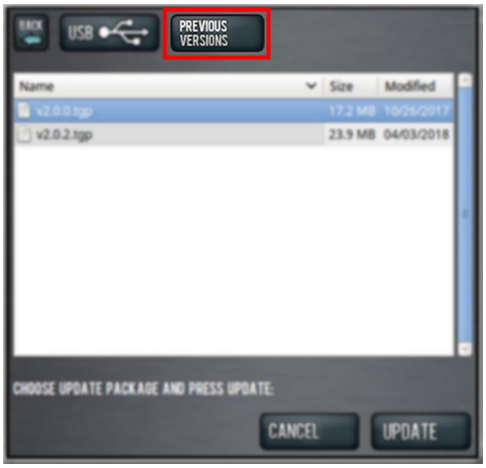


Figure 21: Browse dialog box.

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

PATHPILOT TOOLS AND FEATURES

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

CREATE AND LOAD G-CODE FILES

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

Load G-Code

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

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

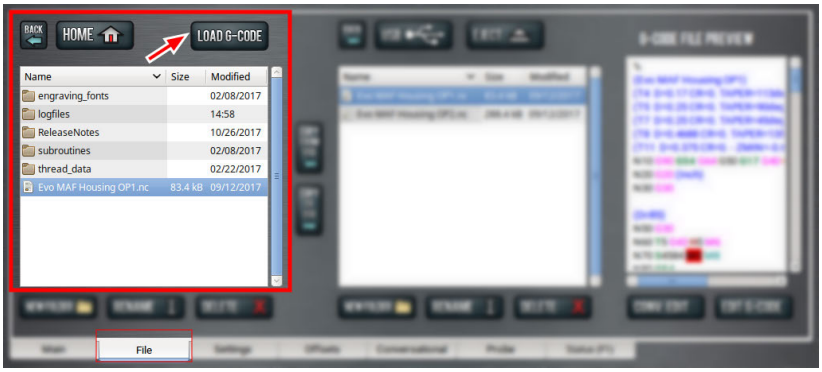


Figure 1: Controller Files window on the File tab.

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

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

Transfer Files to and From the Controller

To run a G-code program, you must transfer the files to the PathPilot controller. For information, see "About System Files" (page 130).
To transfer files to and from the controller:

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

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

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

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

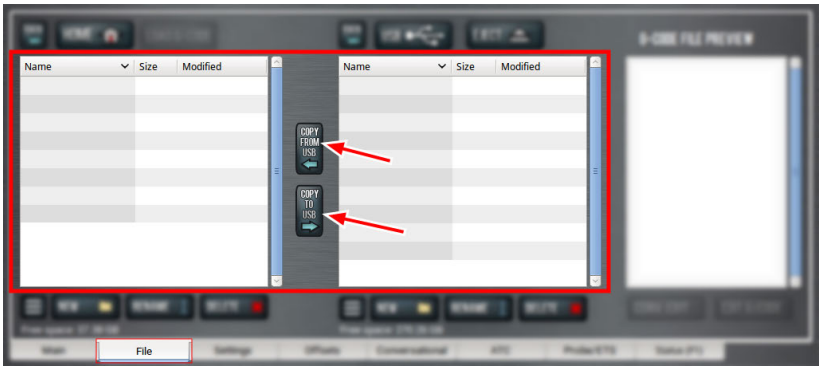


Figure 2: File tab.

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

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

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 3: 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" (page 41).
To access recent G-code files:

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

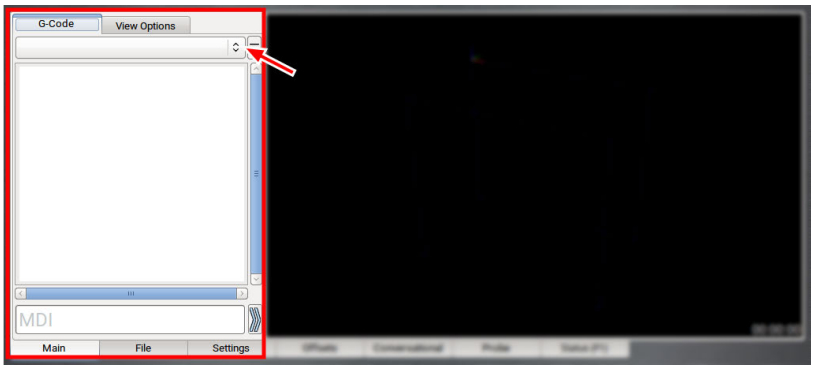


Figure 4: Recent Files menu on the Main tab.

   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**.

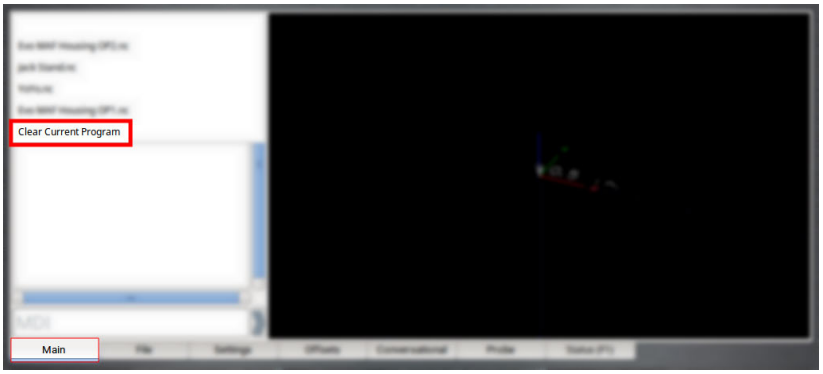


Figure 5: Recent Files menu on the Main tab.

   The currently loaded G-code program closes.

Edit G-Code

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

Edit G-Code with a Text Editor

You can edit .nc files that are on the PathPilot controller. If the .nc file is in the USB Files window, you must first transfer it to the controller; go to "Transfer Files to and From the Controller" (page 35).
To edit G-code with a text editor:

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

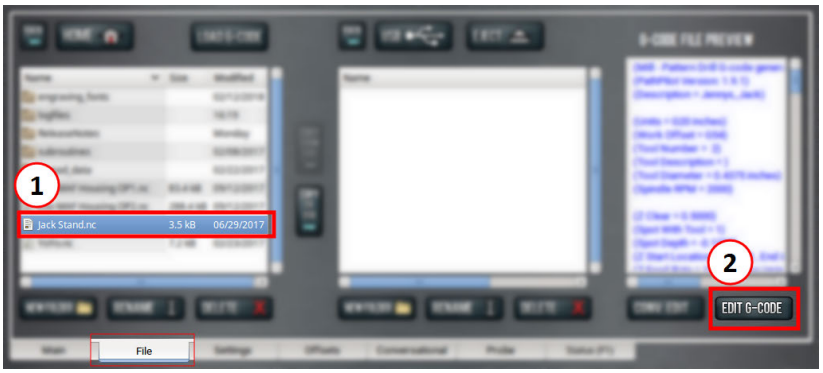


Figure 6: Edit G-code button on the File tab.

   The file opens in a text editor.
2. Make and save the appropriate changes to the file.
3. Close the text editor.

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

Edit G-Code with Conversational Programming

You can edit .nc files that are on the PathPilot controller. If the .nc file is in the USB Files window, you must first transfer it to the controller; go to "Transfer Files to and From the Controller" (page 35).
To edit G-code with conversational programming:

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

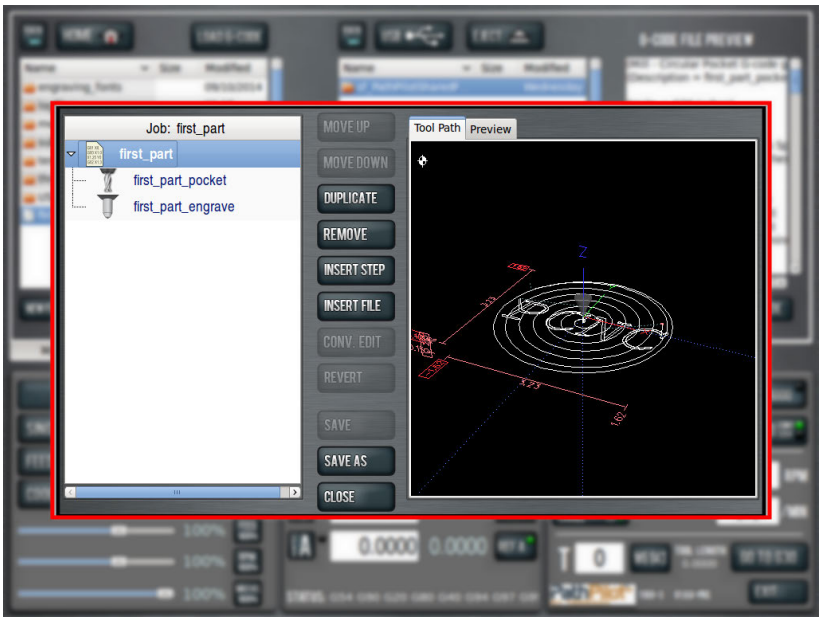


Figure 7: Job assignment editor.

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

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**.

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

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

➢ Select the **Window Expander**.

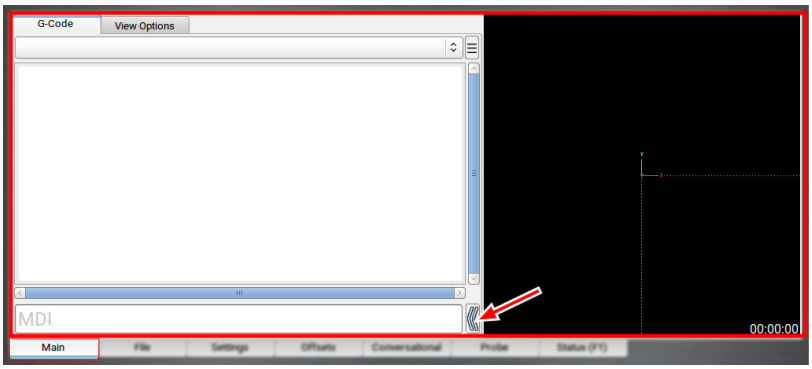


Figure 8: Window Expander on the Main tab.

The **Tool Path** display shrinks.

About the G-Code Tab

The G-Code tab displays the code of the currently loaded program file. Use the scroll bars to view the entire file. You can make the G-Code tab larger. For information, see "Expand the G-Code Tab" (above).
PathPilot highlights certain lines of code of interest. When running a G-code program in single block mode, there may be as many as two lines of G-code highlighted, both with a different color:

- **Green Line** Indicates the start line (the line from which PathPilot starts the program).
  To change the start line, go to "Set a New Start Line" (page 43).
- **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.

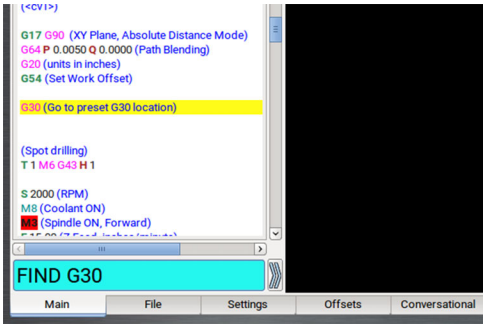


Figure 9: Search for a text command.

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

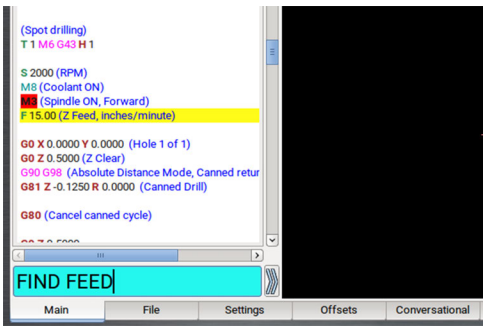


Figure 10: Search for a feed command.

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

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

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

5. *(Optional)* Select **Enter+Shift**.
   PathPilot finds the previous instance of the searched text.

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

Set a New Start Line

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

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

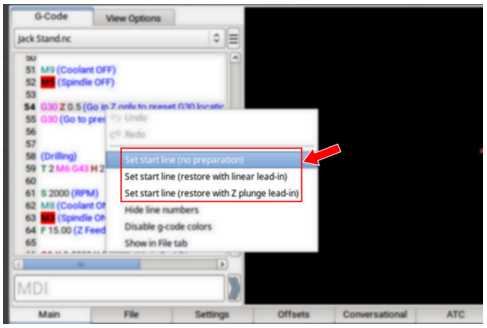


Figure 11: Accessing the Options menu by right-clicking.

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

Lead-In Moves

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

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

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

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

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

  E X A M P L E
  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.

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

Change the View of the Tool Path Display

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

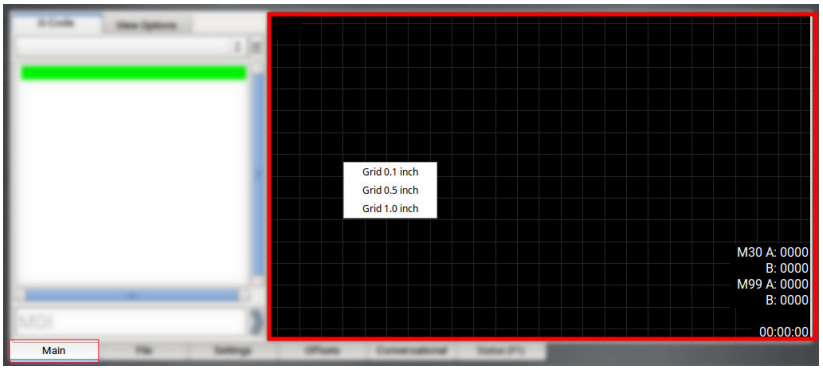


Figure 12: Tool Path display on the Main tab.

- Select the **View Options** tab.

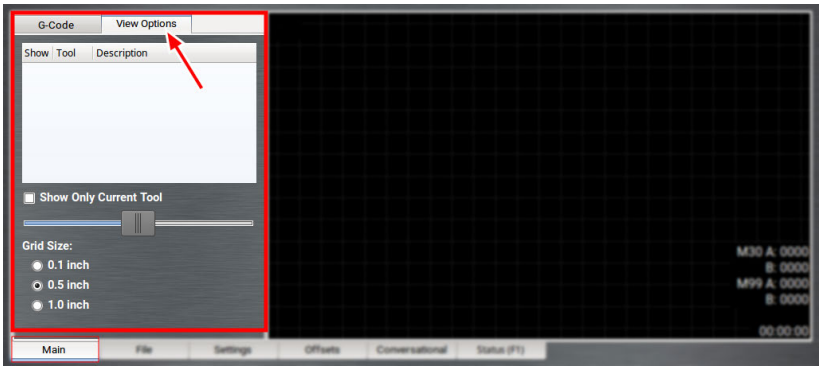


Figure 13: View Options tab on the Main tab.

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

About the Tool Path Display

The Tool Path display is a graphical representation of the currently loaded G-code file's tool path.
Depending on which programming mode you're in (G20 or G21), PathPilot defaults to one of the following grid line spacings:

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

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

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

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

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

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

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

Use Conversational Programming

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

About Conversational Programming

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

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

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

The Conversational tab is divided into two sections:

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

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

- Parameters specific to each operation, like part geometry.

Create an Outside Diameter

Using conversational programming, you can program PathPilot to rough and finish three features: an outside diameter, a fillet (corner radius), or an adjacent face. For information, see "About OD Turning" (page 49).

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

- The value used in the **Z End** DRO field should be less than the value used in the **Z Start** DRO field.
- The value used in the **Filet Radius** DRO field should be larger than the radius of the tool.
- The tool is cutting both an outside diameter and a face — valid tools are limited to orientation 2 for a rear tool, and type 3 for a front tool.
- The face is always on the headstock end of the diameter being cut.
- The fillet calculation doesn't use cutter radius compensation: the middle of the fillet isn't on the true radius for a tool with a tip radius.

To create an outside diameter:

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

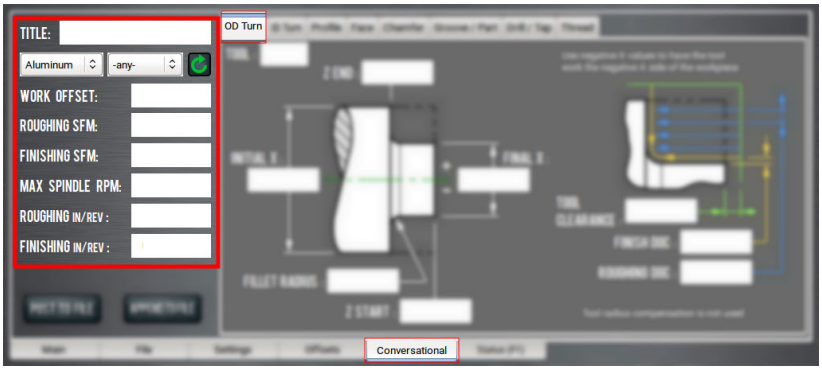


Figure 14: Conversational DROs on the OD Turn tab.

3. Work through the program-specific DRO fields:
   a. In the **Tool** DRO field, type the currently selected tool as it's defined in the **Tool Table** window (on the **Offsets** tab).
   This DRO field is a command value — it sets the tool number for a tool change at the start of the program.
   b. In the **Initial X** DRO field, type the value of the stock's outside diameter.

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

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

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

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

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

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

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

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

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

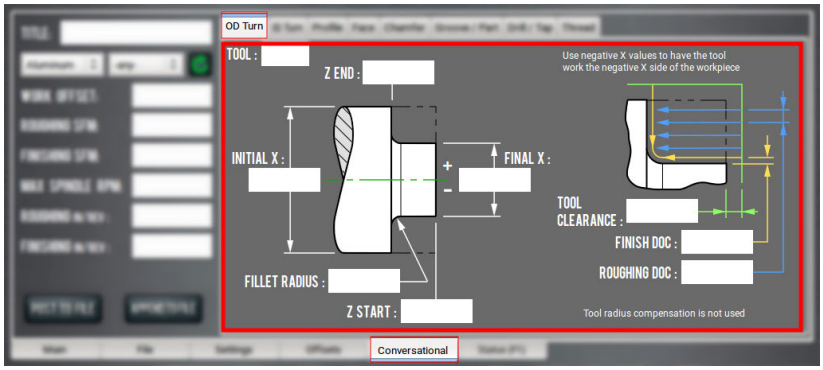


Figure 15: Program-specific DROs on the OD Turn Tab.

About OD Turning

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

OD Turning in PathPilot

During an OD turning routine, PathPilot does the following:

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

   Note: Since there is only one finish pass, the value in the **Finish DOC** DRO field isn't adjusted.
3. The tool retracts to the stock diameter.
4. The face finish pass is cut from the stock diameter to the end of the fillet.

Create an Internal Diameter

Using conversational programming, you can program PathPilot to cut a basic or extended internal diameter on a part. For information, see "About ID Turning" (page 54).

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

- Valid tool orientations are limited to type 2 for a front tool, and type 3 for a rear tool.
- The tool path changes by 90° on the same side of the tool, so a form tool (narrow tip angle) and separate roughing DOCs are needed.

Basic Internal Diameters

To create a basic internal diameter:

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

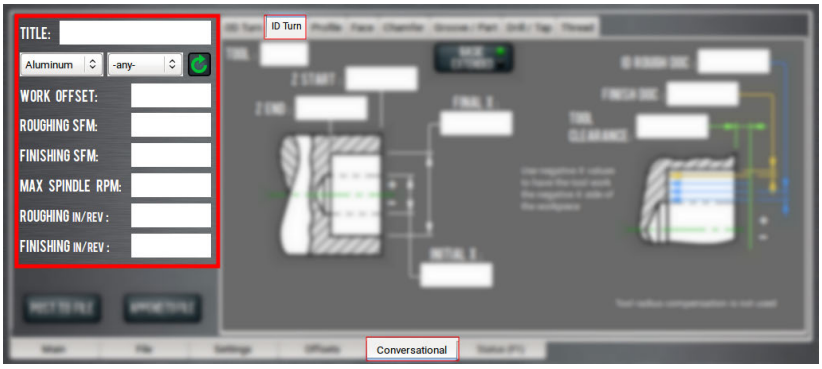


Figure 16: ID Turn tab on the Conversational tab.

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

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

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

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

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

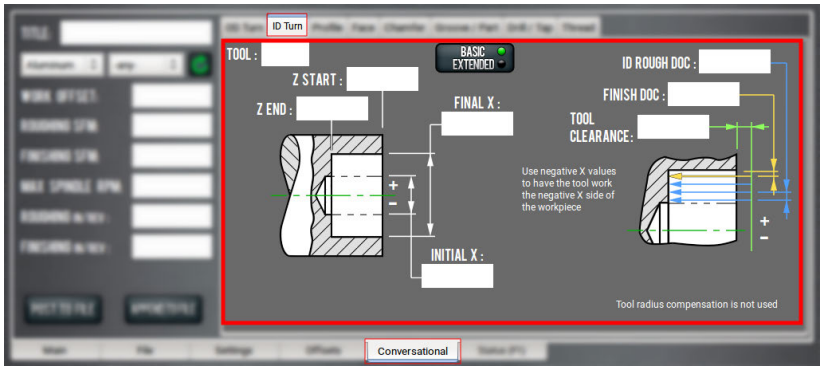


Figure 17: Program-specific DROs on the ID Turn tab.

Extended Internal Diameters

To create an extended internal diameter:

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

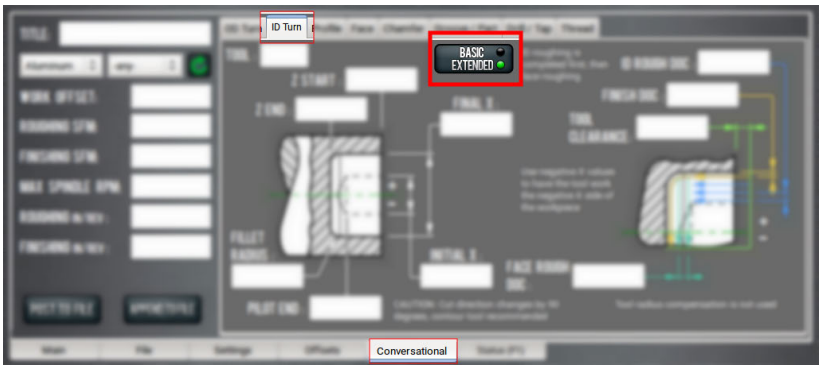


Figure 18: ID Turn tab in Extended mode.

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

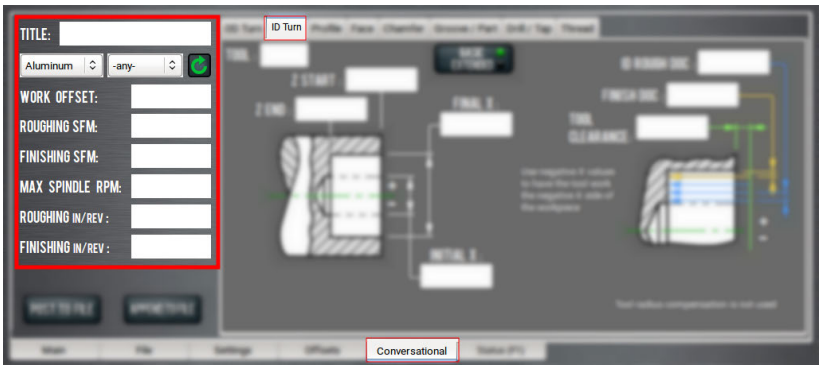


Figure 19: ID Turn tab on the Conversational tab.

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

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

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

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

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

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

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

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

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

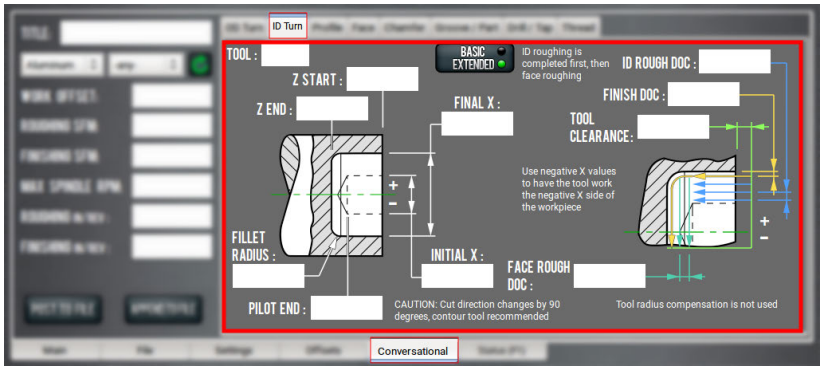


Figure 20: Program-specific DROs on the ID Turn tab.

About ID Turning

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

ID Turning in PathPilot

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

- **Basic Mode**
  Basic mode does one operation, which roughs and finishes from an initial pilot hole diameter to a final internal diameter without cutting a face at the bottom of hole. Use Basic mode for through holes or holes that don't need a finished face. Each pass ends at Z End.
  Roughing starts at the pilot hole diameter (the value in the X Start DRO field), and incrementally cuts diameters with an adjusted depth of cut until the start of the finish diameter (X End - [2 × Finish DOC]). Finishing is done in one pass.
- **Extended Mode**
  Extended mode does three operations: ID roughing, face roughing, and an ID, fillet, and face finish pass. The extended ID roughing passes stop at the bottom of the pilot hole in order to prevent engaging too much of the tool cutting edge. Once the rough ID is cut to the pilot hole bottom, rough facing is started. There are two DRO fields for depth of cut, since, depending on the tool geometry, ID roughing and face roughing may need different depth of cuts.
  For each face pass, the tool tip cuts to the hole center + Tool Clearance which requires a rough hole diameter (which was cut in the first operation) that is a little more than twice the tool's X width. Caution is needed to prevent hitting the back of the tool holder on the side of the hole.

Create a Profile on a Part

Using conversational programming, you can program PathPilot to rough and finish an arbitrary internal or external profile on a part. For information, see "About Profiling" (page 59).

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

- The Tool Clear Dia X DRO field has a value of smaller diameter than first X value on the Profile Point table.
- An internal tool is indicated in the Tool DRO field.

Complete the following steps in the order listed:

Describe the Stock

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

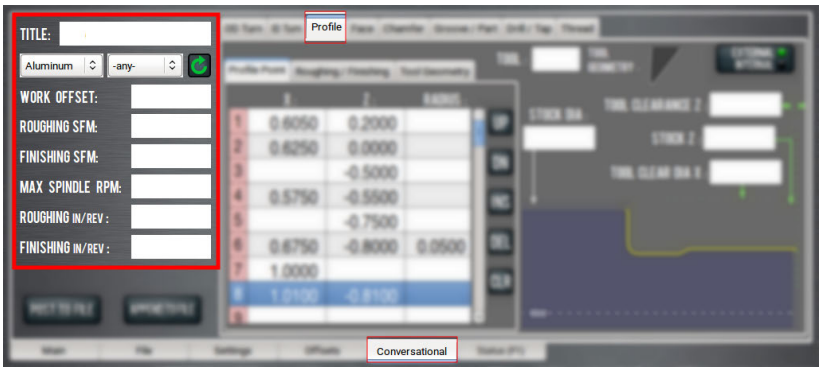


Figure 21: Conversational DROs on the Profile tab.

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

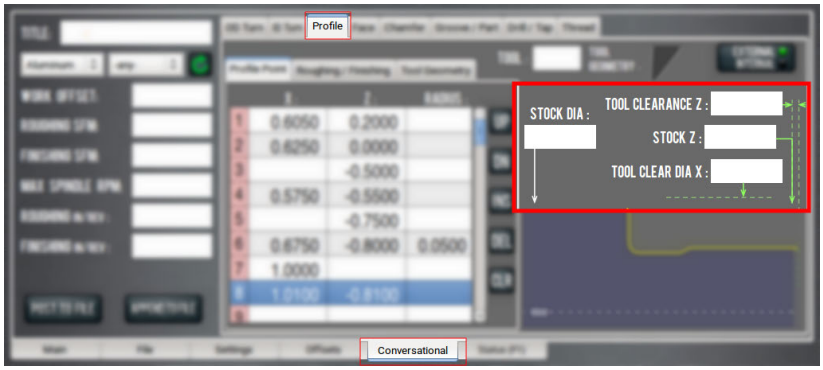


Figure 22: Program-specific DROs on the Profile tab.

Identify the Profile Points

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

1. Select the **Profile Point** tab.
2. In the **Profile Point** table, type the X and Z values for the profile. X values are in diameters terms. The X field is expressed in terms of X+ cutting (X- G-code will properly be generated if the tool selection calls for it).

> **Note:** If the value is unchanged from the previous row in the **Profile Point** table, PathPilot assumes the value is repeated. If you are using the same value, you can leave the cell empty.

3. *(Optional)* In the **Radius** column, type a value to X and Z end points to create an arc.
   a. **For a Center Point Above and/or to the Right of the Start Point** Type a positive radius value.
   b. **For a Center Point Below and/or to the Left of the Start Point** Type a negative radius value.

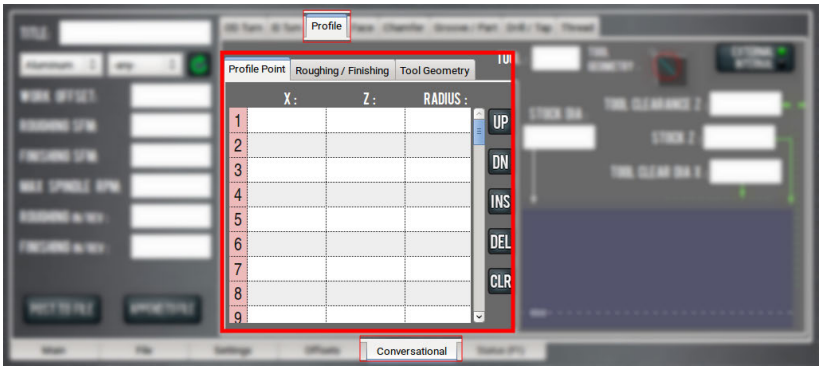


Figure 23: Profile Points table on the Profile tab.

Identify the Tool

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

Identify the Roughing and Finishing

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

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

To identify the roughing and finishing:

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

Note: The default value is 0.02 inches.

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

> **Note:** The default value is 0.003 inches.

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

> **Note:** The default value is 2 (passes).

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

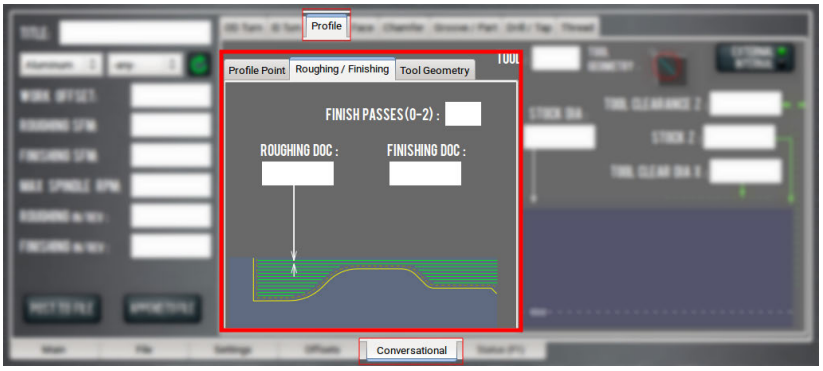


Figure 24: Roughing/Finishing tab on the Profile tab.

Describe the Tool Geometry

Use the Tool Geometry tab to describe the front and rear profiling angles. Because there are many tool and tool holder geometries, the Tool Geometry tab allows you to properly describe each tool and avoid part gouging.
To describe the tool geometry, you must first determine the cutting direction: either X+ or X-, based on the tool orientation and an external or internal profile.
For example, if you are cutting an external profile on a part and have a tool orientation of 2, PathPilot creates X+ G-code.
Use the following table to determine the cutting direction:

Cutting Direction	Internal or External	Tool Orientation
X+ (typically, while using a turret)	External	1, 2, 6
	Internal	3, 4, 8
X- (typically, while using a quick-change tool post)	External	3, 4, 8
	Internal	1, 2, 6

To describe the tool geometry:

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

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

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

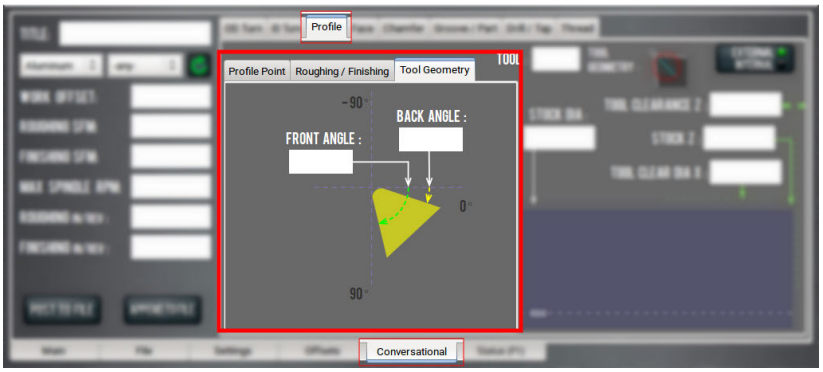


Figure 25: Tool Geometry tab on the Profile tab.

About Profiling

The profile is created from a list of points that describes the part geometry. A profile can have both forward (toward the tailstock) and rear (toward the headstock) facing features, and can also start behind the highest Z plane (the feature that is closest to the tailstock).
On the Profile tab, you can also specify things like:

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

Create a Face on a Part

Using conversational programming, you can program PathPilot to cut a face with tool paths from the stock's outer diameter to the spindle center or an inner diameter (with each pass progressing in Z toward the headstock). For information, see "About Facing" (page 61).

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

- To cut with a rear tool, the values used in the **Initial X** DRO field and the **Final X** must be positive. The tool works on the positive X side of the spindle center (the side away from you).
- To cut with a front tool, the values used in the **Initial X** DRO field and the **Final X** must be negative. The tool works on the negative side of the spindle (the side closest to you).

- The value used in the **Roughing DOC** DRO field must be positive.
- The value used in the **Finish DOC** DRO field must be positive.
- Spindle speed control: CSS.
- Feed rate control: FPR.

To create a face on a part:

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

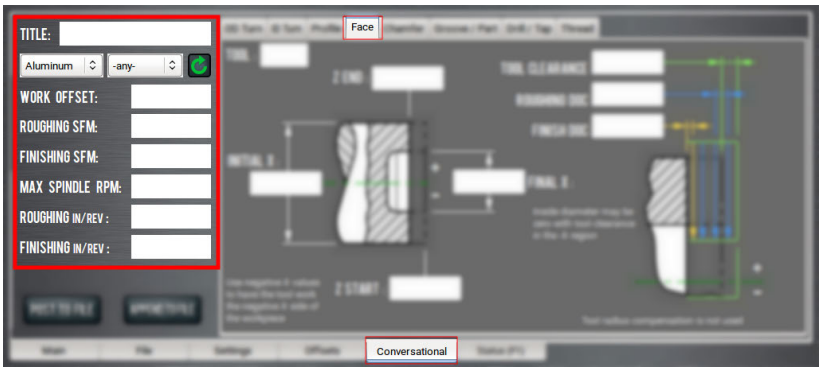


Figure 26: Conversational DROs on the Face tab.

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

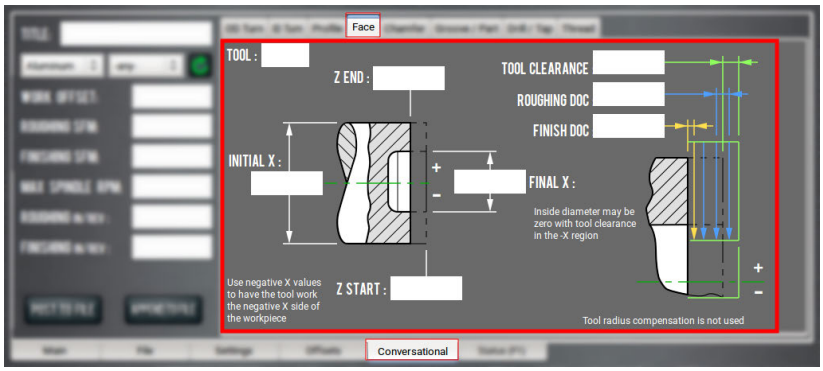


Figure 27: Program DROs on the Face tab.

About Facing

During a facing routine, PathPilot does the following:

1. Rough facing starts at Z Start and incrementally cuts at the depth of cut until the start of the finish face pass (Z End + Finish DOC).

2. The start of each pass is at the Initial X diameter + Tool Clearance and moves in the minus X direction until the Final X diameter – Tool Clearance is reached.
   If the value in the Final X DRO field is zero, the end of the pass will go beyond the spindle center.
3. Finishing is done in one pass at the value entered into the Finish DOC DRO field.

Create a Chamfer or Radius on a Part

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

Before You Begin

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

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

To create a chamfer or radius on a part:

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

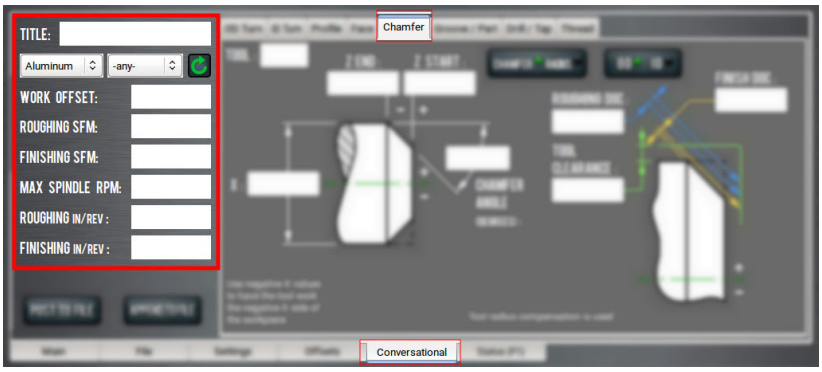


Figure 28: Conversational DROs on the Chamfer tab.

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

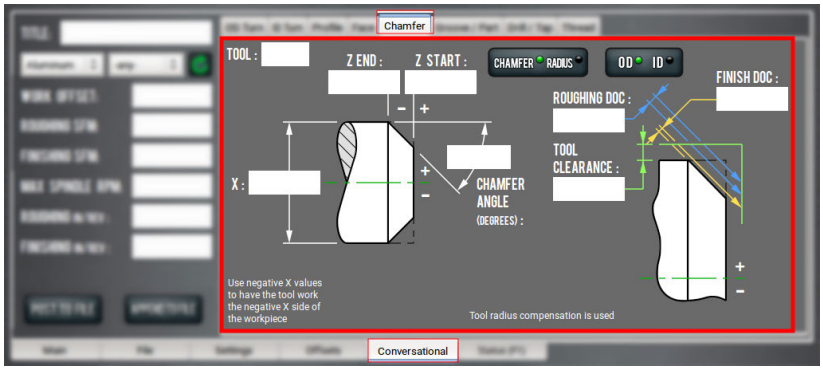


Figure 29: Program DROs on the Chamfer tab.

About Chamfer and Radius

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

1. Roughing starts at the corner of X and Z Start in adjusted depth of cut increments perpendicular to the chamfer angle or incremental arcs for radius.
2. The last roughing pass leaves enough material for the finish pass; finishing is done with a single pass.

3. Passes start and end on the perimeter of the tool clearance space, which is set by adding the tool clearance DRO value to the stock OD, X, and the face location, Z Start.

Create a Groove or Part a Workpiece

Using conversational programming, you can program PathPilot to create a groove or to part a workpiece from stock. For information, see "About Grooving and Parting" (page 67).

Before You Begin

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

- Grooving paths are based on **Z Start** and **Z End** values:
  - If the value in the **Z Start** DRO field is greater than the value in the **Z End** DRO field, the tool's control point is set to the +Z side of the tool.
  - If the value in the **Z Start** DRO field is less than the value in the **Z End** DRO field, the control point is set to the -Z side of the tool.
- Groove roughing is done with plunge cuts in the X direction. Each plunge is incremented in the Z direction from **Z Start** ± **Finish DOC** to **Z End** ± (**Tool Width** + **Finish DOC**).
- Even though a grooving/parting tool may be considered to have two tips, valid tool orientation is limited to:
  - Groove on the negative side of Z Start, Back Tool = Type 1
  - Groove on the positive side of Z Start, Back Tool = Type 2
  - Groove on the positive side of Z Start, Front Tool = Type 3
  - Groove on the negative side of Z Start, Front Tool = Type 4
  - Part on the negative side of Z Start, Back Tool = Type 1
  - Part on the negative side of Z Start, Front Tool = Type 4
- CSS is used for spindle speed control.
- FPR is used for feed rate control.
- CRC is not used.

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

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

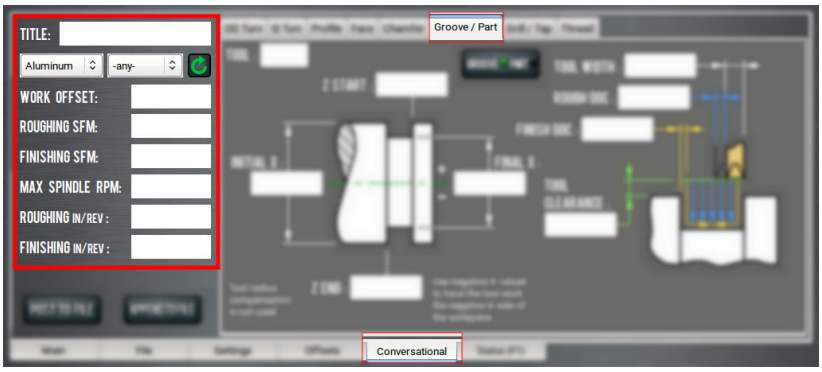


Figure 30: Conversational DROs on the Groove/Part tab.

3. If required, toggle the **Groove/Part** button. Then, do one of the following:
   - Go to "Create a Groove on a Part" (on the next page).
   - Go to "Part a Workpiece from the Stock" (page 67).

Create a Groove on a Part

Work through the program-specific DRO fields:

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

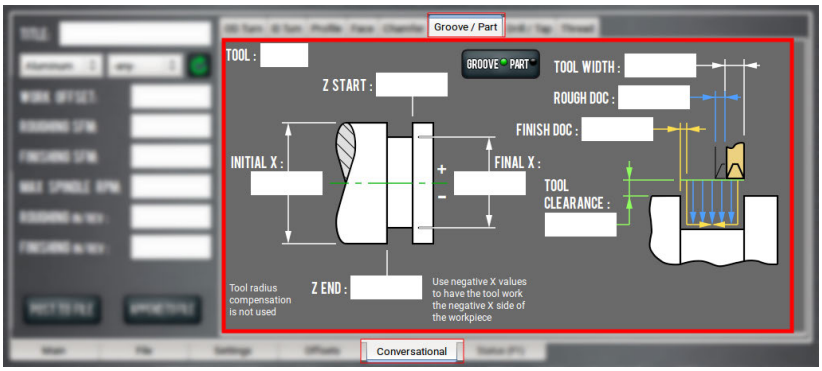


Figure 31: Program-specific DROs on the Groove/Part tab.

Part a Workpiece from the Stock

Work through the program-specific DRO fields:

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

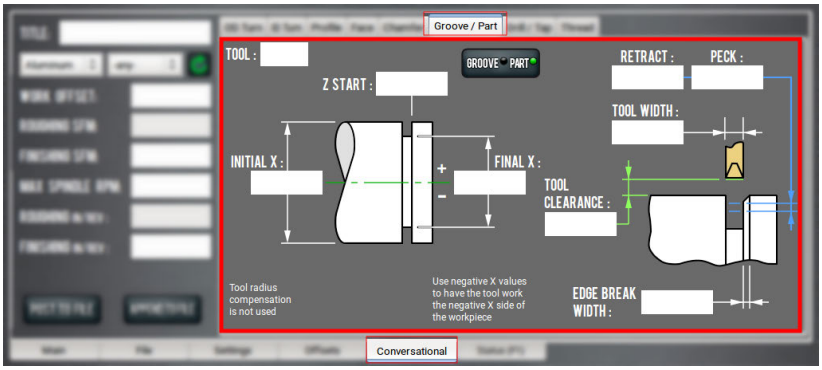


Figure 32: Program-specific DROs on the Groove/Part tab.

About Grooving and Parting

1. Groove finishing is done with a plunge cut down the Z End side.
2. When the tool reaches the bottom, the tool is moved in the Z direction toward the center of the groove, then retracts.
3. The tool is plunged on the Z Start side of the groove, then again is moved in Z toward the groove center and retracted. This requires a grooving tool, which can side cut. Part does one plunge cut at Z Start. The tool's control point is on the +Z side of the tool. The plunge cannot be set to go beyond the spindle center (X = 0).

Create Holes on a Part or Tap a Hole

Using conversational programming, you can program PathPilot to drill holes on a part, or use rigid tapping to thread holes on a part. For information, see "About Drilling and Tapping" (page 70).

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

- The value used in the **Z Start** DRO field must be larger than the value in the **Z End** DRO field.
- For tapping, the value used in the **Z End** DRO should allow for extra threading while the spindle comes to a stop and reverses.
- The value used in the **Peck Depth** DRO field needs a direction, so should have a negative value.
- Drilling is limited to the -Z direction, toward the spindle.
- Use the **RPM DRO** field instead of CSS.

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

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

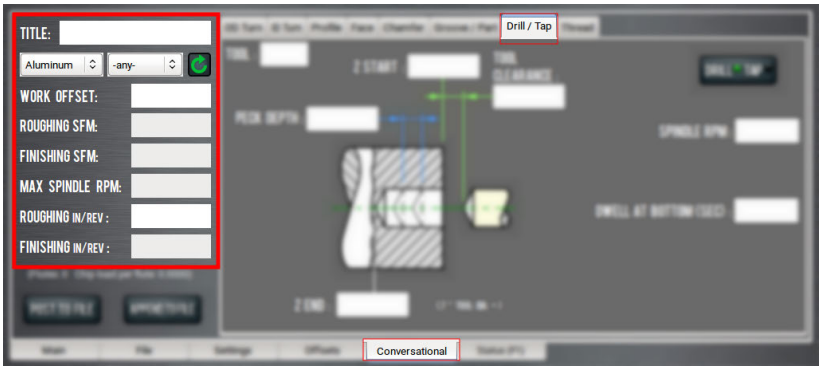


Figure 33: Conversational DROs on the Drill/Tap tab.

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

Create a Hole on a Part

Work through the program-specific DRO fields:

1. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
2. In the **Z Start** DRO field, type the stock face location. This field is also used with the **Tool Clearance** DRO field to set the transition between rapid and the feed for drilling or tapping.
3. In the **Z End** DRO field, type the final depth. This is the location where the drill feed stops and optionally dwells.
4. In the **Peck Depth** DRO field, type an incremental depth for retracting the drill to clear chips from the hole, if required. If drilling the hole doesn't need a peck, type `0`. To make each peck depth equal, the value is adjusted to fit an integer number of pecks within the hole depth.

5. In the **Tool Clearance** DRO field, type the desired space required for tool retraction and transitions between rapid and cutting feed rate.
6. In the **Spindle RPM** DRO field, type the RPM (G97) desired.
7. In the **Dwell at Bottom (Sec)** DRO field, type the length of time that Z motion should pause so that the drill can finish cutting the hole bottom before retracting.

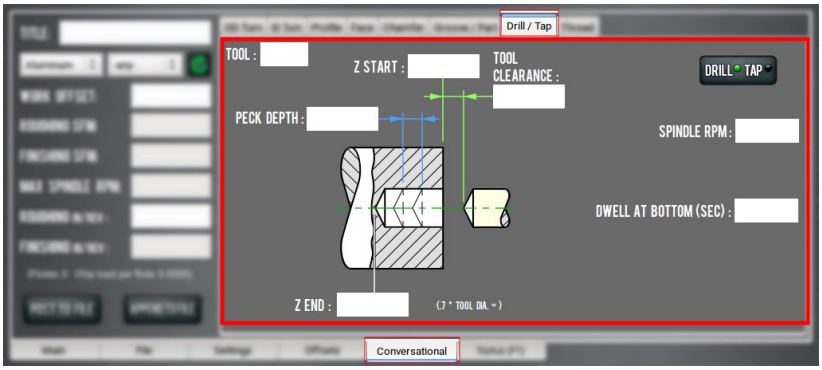


Figure 34: Program-specific DROs on the Drill/Tap tab.

Create Threads in a Hole

Work through the program-specific DRO fields:

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

8. In the **Pitch (Inches)** DRO field, type the Z motion to spindle ratio that matches the required thread pitch. This field is also used with the **Threads/Inch (/mm)** DRO field, so entering a value in either field calculates and inserts a value in the other.

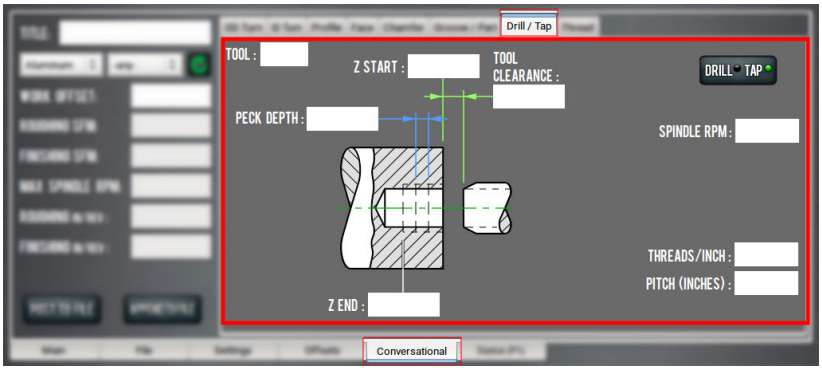


Figure 35: Program-specific DROs on the Drill/Tap tab.

About Drilling and Tapping

Drilling

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

Dwell allows a pause for the drill to stay at Z End long enough to cut a full revolution at the bottom of the hole (rather than immediately retracting the drill, which could leave an irregular bottom).
Pecking can help clear chips before they bind in the hole during drilling. The peck motion retracts to Z Start + Tool Clearance on each cycle.

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

Tappinsg

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

1. Tapping starts with the tap at Z Start + Tool Clearance.
2. Z motion waits until the spindle encoder index is tripped. Then, the gears are engaged and Z feeds at the rate set by the pitch or threads per unit (TPU) and spindle encoder count. The Z motion follows the spindle motion no matter what the spindle does.

3. For tapping, the spindle is run forward until Z End is reached, the spindle is reversed, which causes it to slow to a stop, then reverse. During this time, the tap continues to follow the spindle motion and continues to make threads until the spindle reverses.

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

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

Create Threads on a Part

Using conversational programming, you can program PathPilot to single point an external or an internal thread on an existing outer diameter. For information, see "About Threading" (page 73).

Before You Begin

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

- The value used in the **Z Start** DRO field must be larger than the value used in the **Z End** DRO field.
- The value used in the **Depth of Cut** DRO field must be positive.

To create threads on a part:

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

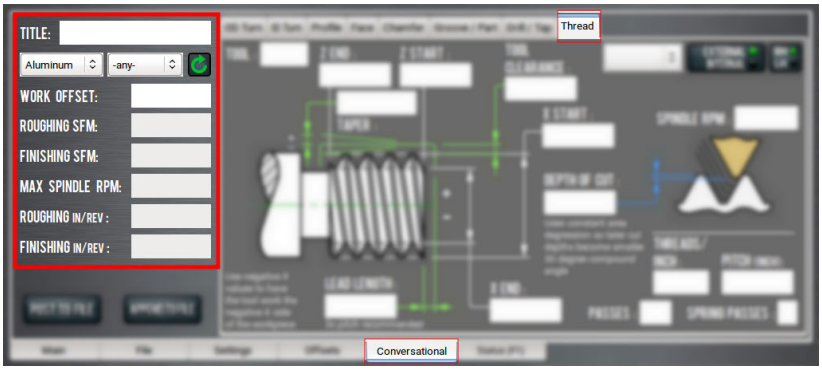


Figure 36: Conversational DROs on the Thread tab.

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

4. Work through the program-specific DRO fields:
   a. In the **Tool** DRO field, type the tool number for use with the program. This sets the tool number for a tool change at the start of the program.
   b. In the **X Start** DRO field, type the existing major diameter for external threads, or minor diameter for internal threads.
   c. In the **X End** DRO field, type the location of the new outside diameter.
   d. In the **Z Start** DRO field, type the stock face location. This value is also used with the Tool Clearance DRO field to set the transition between rapid and feed rate on some Z moves.
   e. In the **Z End** DRO field, type the final thread Z location.
   f. In the **Tool Clearance** DRO field, type the desired space beyond the stock outside diameter that's required for rapid movements to clear the workpiece. This sets the starting X diameter of the thread cycle return path. Larger values may be safer; smaller values may save time once you're familiar with how well the program works.
   g. In the **Lead Length** DRO field, type the required length during the start of the cutting path that allows the motion to stabilize before cutting material. The start of a cutting pass waits for the spindle encoder index to trip; when it does, the Z motion tries to instantly match the spindle speed, but actually needs time to accelerate and match the spindle encoder count. This value must be a compromise between spindle speed, thread pitch, and workpiece clearance.
   h. From the **Thread Table** drop-down menu, select the value for the thread. The threads listed in this menu follow the current unit setting (inch or millimeter). Once you make a selection, the data for that thread is entered into the appropriate DRO fields.

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

   i. In the **Spindle RPM** DRO field, type the spindle rpm.
   j. In the **Depth of Cut** DRO field, type the depth of material being cut. Each pass is incremented based on a calculation of the area of the material being removed. This allows for a constant tool load for each pass.
   k. In the **Threads/Inch (/mm)** DRO field, type the number of threads per inch (or millimeter).
   l. In the **Pitch (Inch)** DRO field, type the distance per thread.
   m. In the **Passes** DRO field, type the number of successive passes to cut the thread.

n. In the **Spring Passes** DRO field, type the number of extra passes at full thread depth. We suggest using this to clean up the thread and compensate for any material deflection during thread cutting.

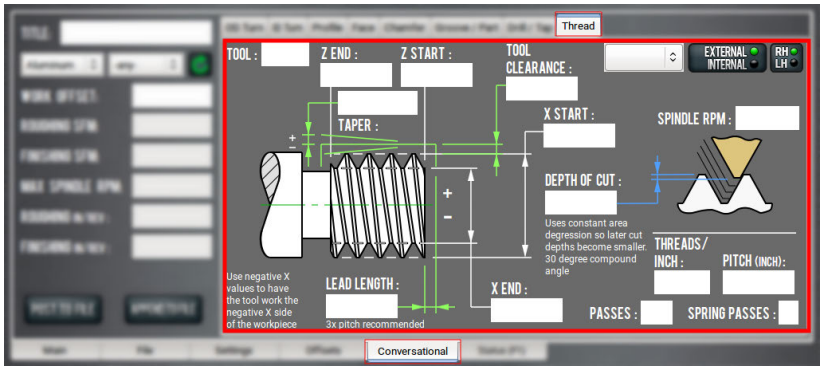


Figure 37: Program-specific DROs on the Thread tab.

About Threading

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

MACHINE SETTINGS AND ACCESSORIES

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

Enable an Internet Connection

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

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

Figure 38: Internet button on the Settings tab.

The **Network Configuration** dialog box displays.

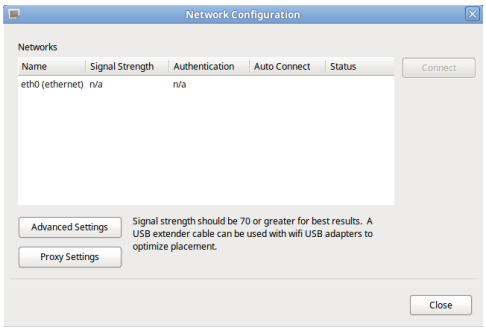


Figure 39: Network Configuration dialog box.

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

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

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

3. Once connected, you can use the Dropbox and automatic updates features. Depending on what you want, see the following procedures:
   - "Connect to Dropbox" (below)
   - "Enable Automatic Updates" (page 77)

Connect to Dropbox

Note: Dropbox requires an internet connection. If you haven't yet enabled it, go to "Enable an Internet Connection" (on the previous page).

If desired, you can connect your PathPilot controller to a Dropbox account to easily synchronize your G-code files, which eliminates the need to transfer them with a USB drive.

Note: Dropbox stops synchronizing once the PathPilot controller's internal drive has less than 500 MB of free space. To avoid this, we recommend that you organize your Dropbox account on a separate computer before you connect to Dropbox with PathPilot. Only store files in the top-level that you want synchronized to your PathPilot controller.

To connect to Dropbox:

1. From the PathPilot interface, on the **Main** tab, in the **MDI Line** DRO field, type `ADMIN DROPBOX`.
2. Select the **Enter** key.
   The **Dropbox Configuration** application displays.

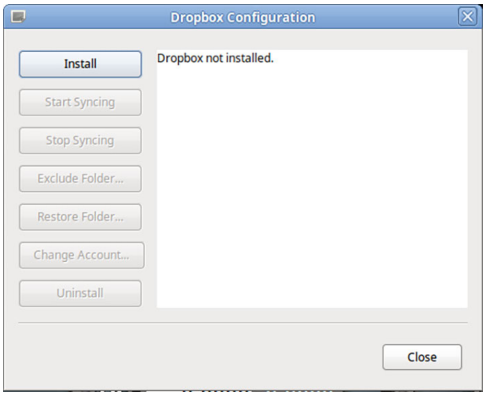


Figure 40: Dropbox Configuration application.

3. Select **Install**.
   The **Dropbox Installation** dialog box displays.
4. Select **OK**.

Figure 41: Dropbox Installation dialog box.

   The Dropbox installation starts and continues for about a minute. When done, a web browser displays.
5. From the web browser, sign in or create a Dropbox account.
   The PathPilot controller connects to the Dropbox account, creates a local **Dropbox** folder that is visible in **File** tab, and synchronizes the folder.

6. Because Dropbox stops synchronizing once the PathPilot controller's internal drive has less than 500 MB of free space, we recommend that you exclude large or unrelated folders from synchronization. Select **Exclude Folder...**.

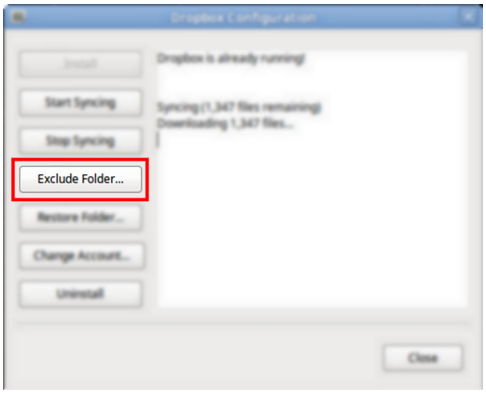


Figure 42: Exclude Folder button.

The **Add Folder to Excluded Set** dialog box displays.

7. In the **Add Folder to Excluded Set** dialog box, type the name of the folder to exclude.

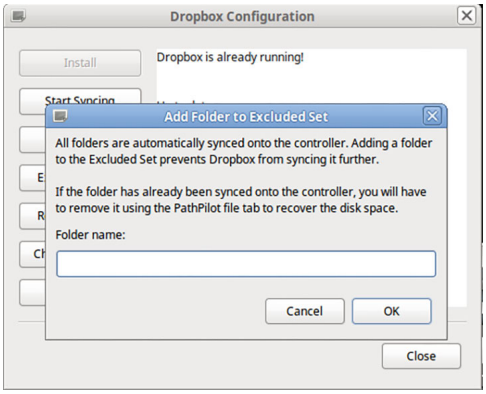


Figure 43: Add Folder to Excluded Set dialog box.

> **Note:** You can only exclude folders, not individual files.

8. Select **OK**.
9. Select **Close**.
   The PathPilot controller is now connected to Dropbox.

Enable Automatic Updates

> **Note:** Automatic updates require an internet connection. If you haven't yet enabled it, go to "Enable an Internet Connection" (page 74).

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.

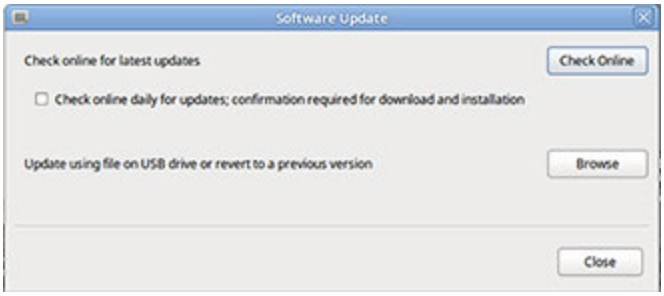


Figure 44: Software Update dialog box.

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.

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.

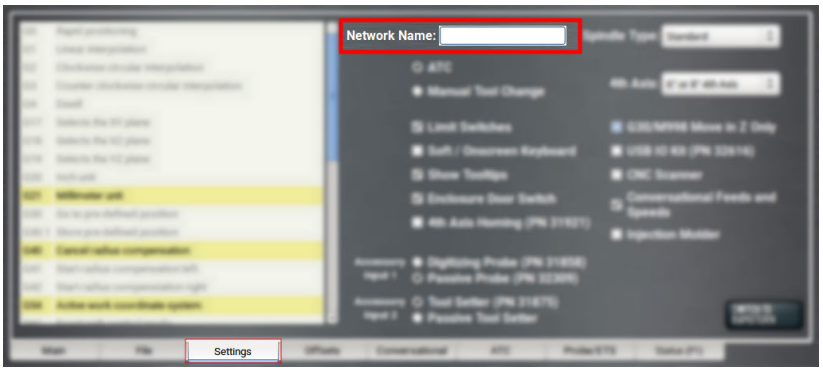


Figure 45: Network Name field on the Settings tab.

> **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.

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" (on the next page).

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.

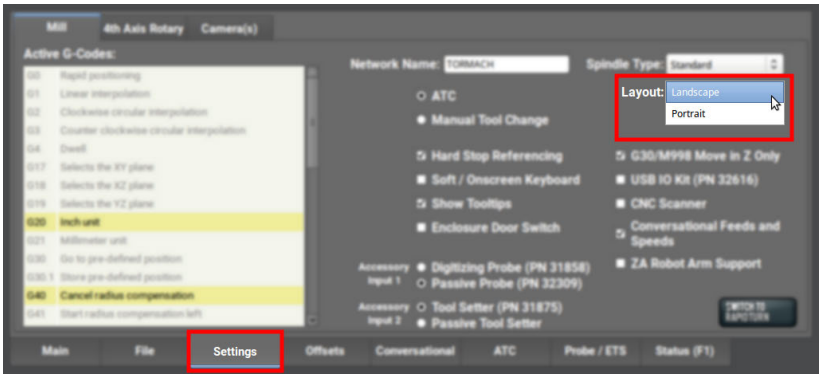


Figure 46: Layout drop-down menu on the Settings tab.

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.

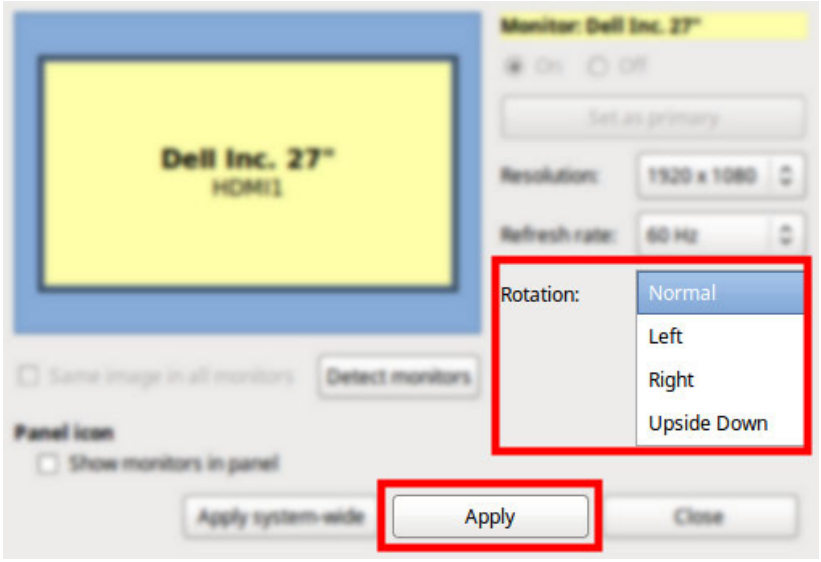


Figure 47: Monitor configuration dialog box.

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.

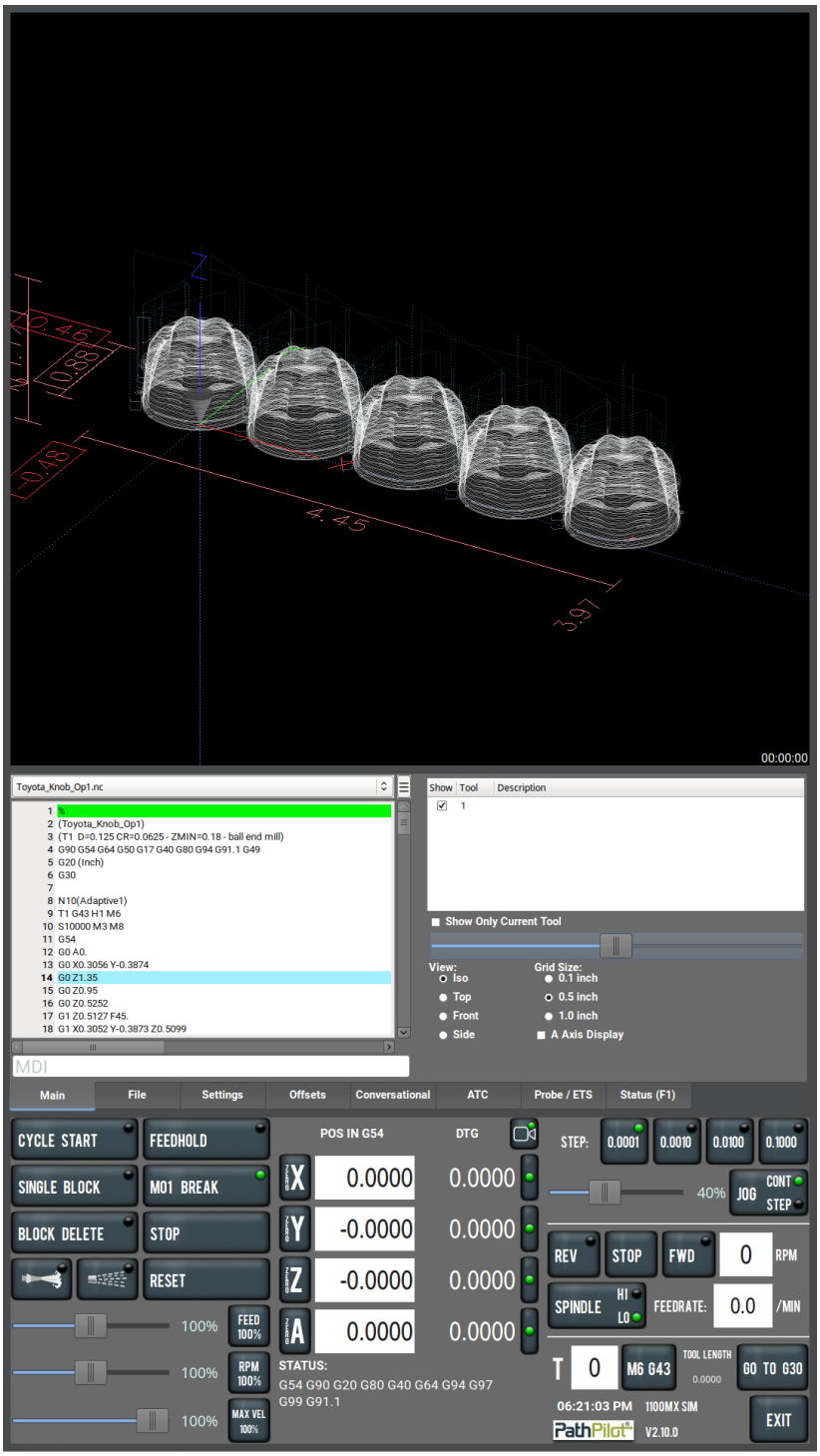


Figure 48: Tool Path window in portrait screen layout.

- 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.

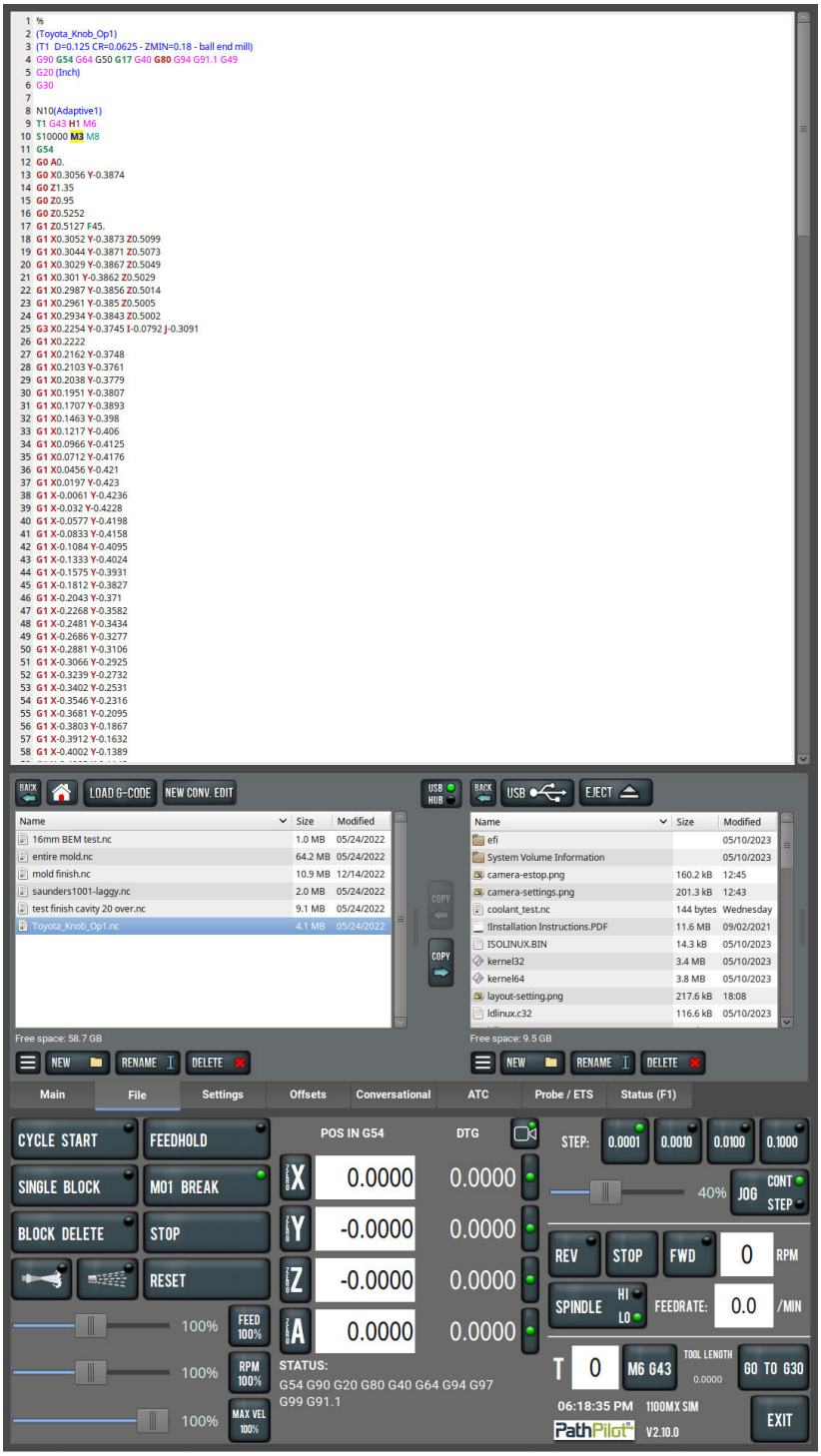


Figure 49: File tab G-code preview in portrait screen layout.

Specify the Spindle Pulley Option

➢ From the Settings tab, select one of the following:

- **Collet (High Speed)**
- **Chuck (Low Speed)**

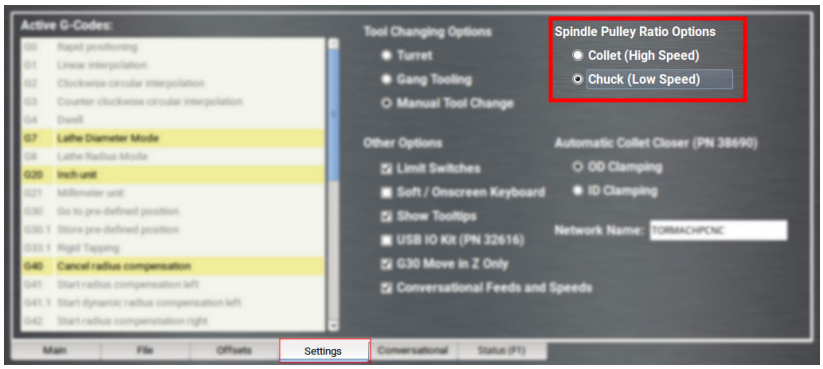


Figure 50: Spindle pulley options on the Settings tab.

Specify the Tool Change Method

When PathPilot finds an `M06` command in a G-code program, it has different behaviors depending on the specified tool change method.
To specify the tool change method:

➢ From the **Settings** tab, select the appropriate tool change method for your machine.

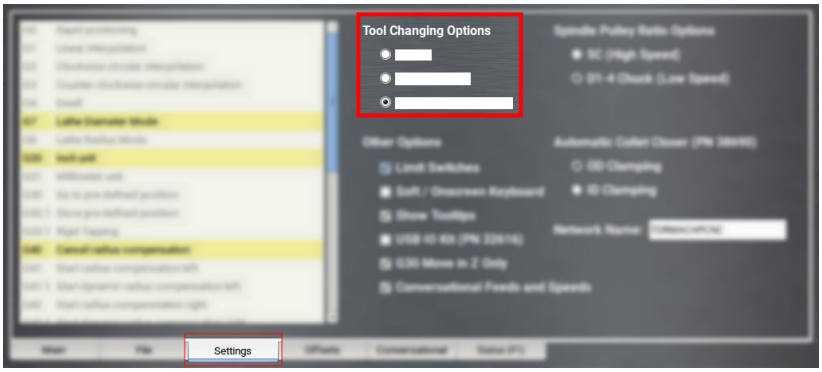


Figure 51: Tool change method options on the Settings tab.

About Turret Tool Changes

T commands T01 through T08 cause the turret to index (regardless of current machine position) and the offsets for the requested tool are applied. With the turret setup, tool numbers higher than eight are still available, and a T command of, for example, T09 does not cause turret rotation but does apply the offsets for tool 9. It is possible to mount a parting tool or gang of drills to the lower portion of the carriage and assign tool numbers higher than eight to these tools.
The machine does not automatically retract to G28 or G30 position before a tool change. You must program the machine to a safe position before executing a T command. If generating code using the conversational features of the control, a G30 is inserted before each tool change command.

About Gang Tool Changes

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

About Manual Tool Changes

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

About Mixed Tooling Changes

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

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

   Note: We recommend this option.

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

Disable Limit Switches

To provide a temporary workaround for a malfunctioning limit switch circuit, you can disable the limit switches. For information, see About Limit Switches.

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

To disable limit switches:

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

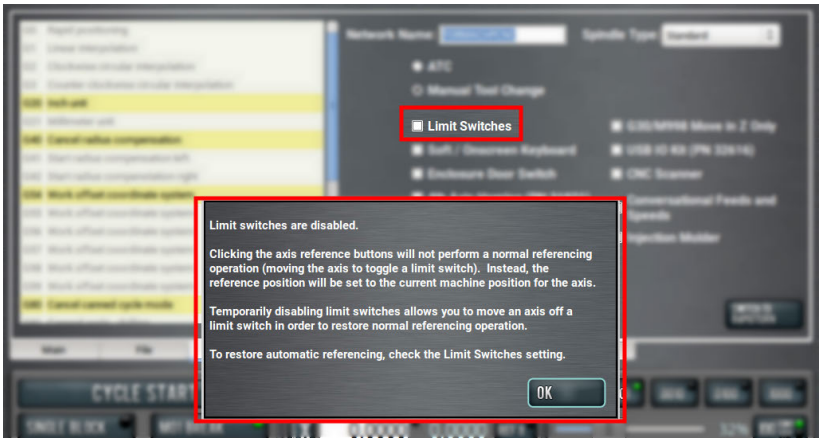


Figure 52: Limit Switches checkbox on the Status tab.

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.

Limit G30 Moves

You can limit G30 moves so that only the Z-axis moves. For information, see "About G30" (page 120).
To limit G30 moves:

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

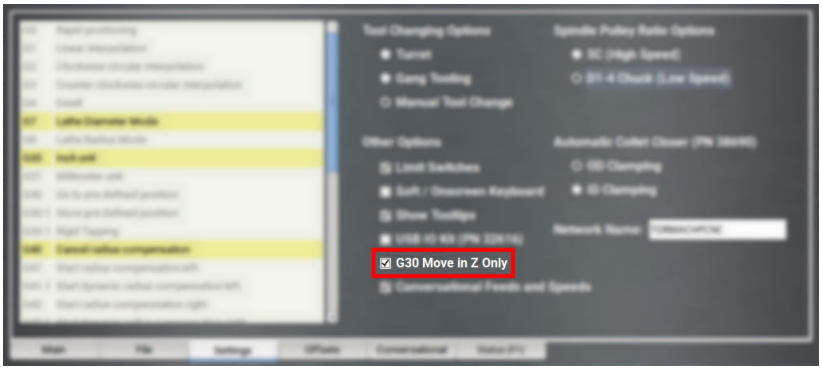


Figure 53: Settings tab.

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 119).
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.

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**.

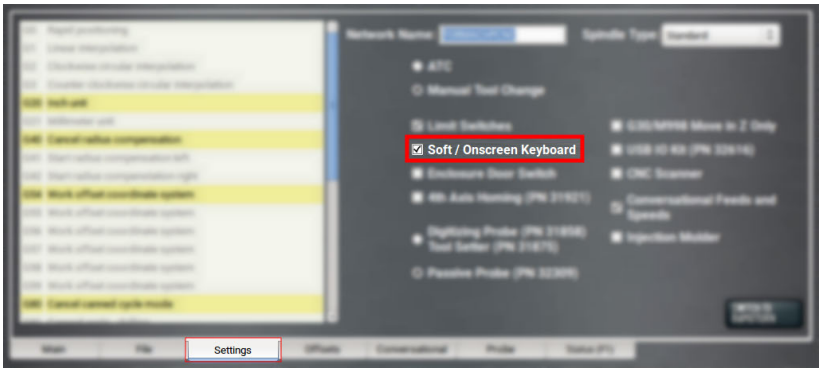


Figure 54: Settings tab.

2. To resize the keyboard, select a corner of the keyboard and drag.
3. To reposition the keyboard, select the **Anchor** key and drag the keyboard anywhere on the screen.

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

About Soft Keyboards

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

Figure 55: Soft (on-screen) keyboard.

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)**.

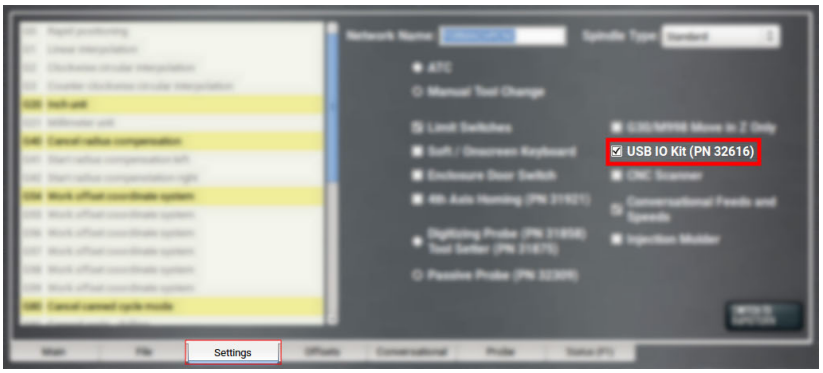


Figure 56: Settings tab.

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**.

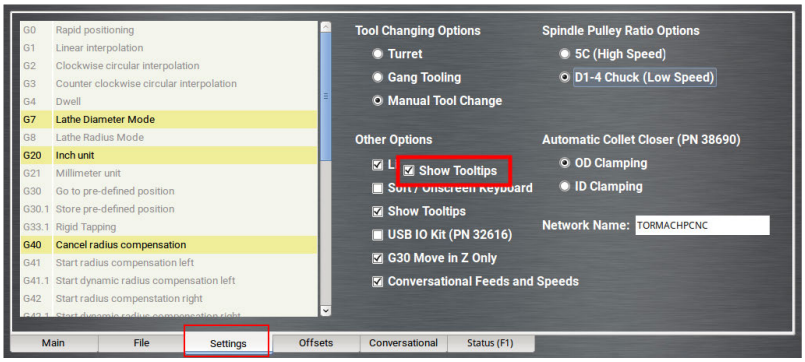


Figure 57: Show Tooltips checkbox.

> **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.

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 122).

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

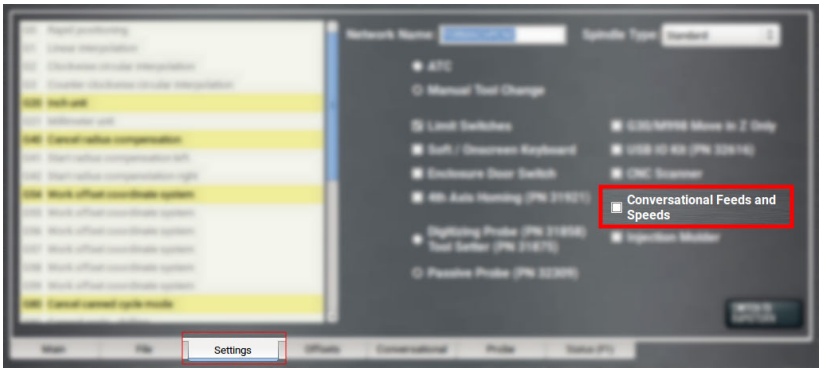


Figure 58: Settings tab.

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.

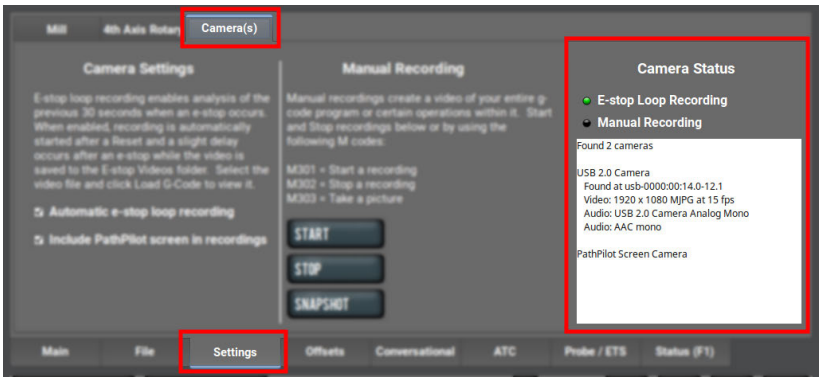


Figure 59: USB camera status.

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.

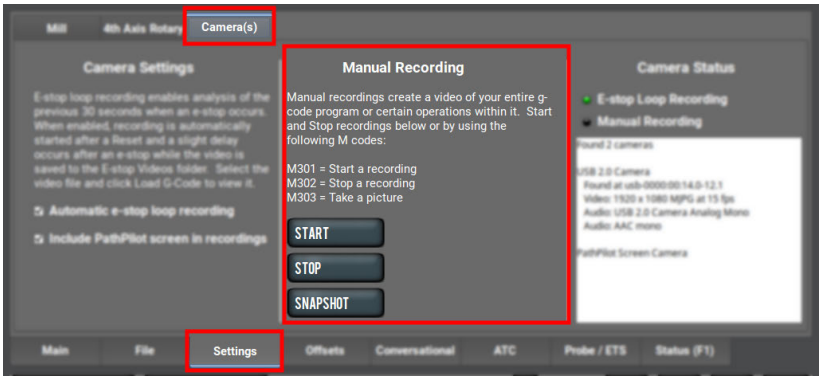


Figure 60: Manual recording controls.

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

Figure 61: Video Camera Recording button.

Whenever PathPilot is recording from a USB camera and/or the virtual screen camera, the LED on this button is green. If PathPilot is recording and the button is pressed:

- If a program is running and not paused at an M00/M01, the recording is aborted.
- If a program is not running, but the machine is moving, the recording is aborted.
- Otherwise, if a manual recording is in progress, it is stopped and a file save as dialog will appear. If an automatic e-stop loop recording is in progress, it is aborted since no e-stop occurred.

To include a screen recording:

1. Toggle the **Include PathPilot screen in recordings** checkbox in the **Camera Settings** area of the **Camera(s)** tab to enable or disable screen recording.

Figure 62: 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.

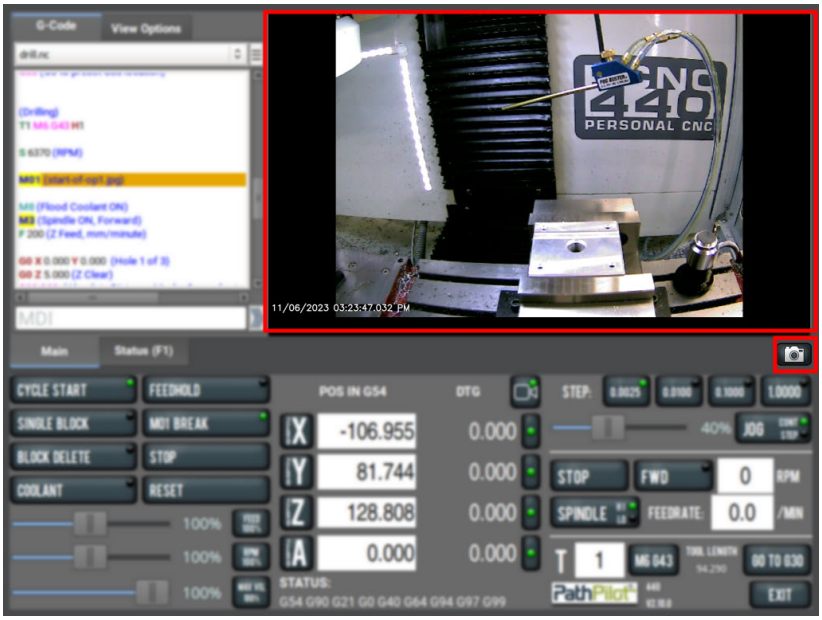


Figure 63: Example of taking a photo.

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 171).

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.

Set Tool Geometry Offsets

Before running a G-code program, PathPilot must know the geometry of the tools that are required for the program. For more information on using tool length offsets, see "About Tool Offsets" (page 95).

Note: You can import a .csv file with tool geometry offset data. For information, see "Import and Export the Tool Table" (page 134).

To set tool geometry offsets:

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

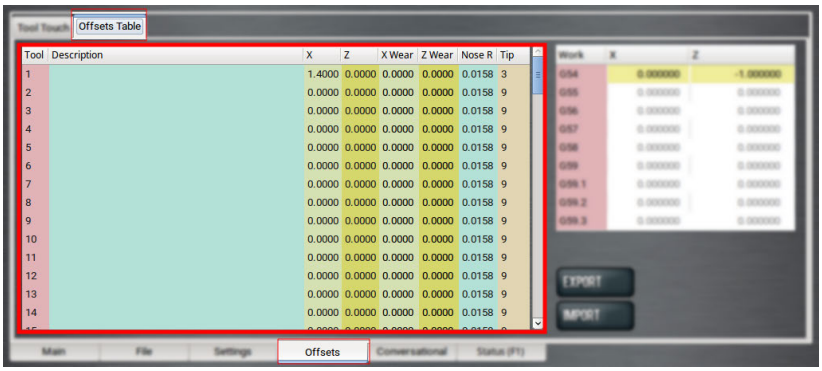


Figure 64: Tool Table window on the Offsets tab.

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

Touch Off the Tool Geometry Offsets

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

Tip! When using this method to measure your tool geometry, remember that the X zero location never changes (the spindle centerline is always X = 0), but the Z zero location may change depending on the length of the workpiece that is chucked into the spindle. As long as each tool is

measured to a face that has been zeroed, only measure these tools one time or until you replace an insert.

To touch off the tool geometry offsets:

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

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

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

Touch X

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

   Note: If you're touching off a front tool post tool, verify that the value you enter is negativepositive.
5. Select **Touch X**.
   The LED comes on.

Touch Z

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

About Tool Offsets

Tool offsets allow you to use various tools while still programming with respect to the workpiece. Tools can have different lengths (and, while using gang tooling, different X/Z positions on the carriage).
Tool offsets are broken down into two components:

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

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

- **Wear Offsets**

The sign convention for the machine are as follows:

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

Sign convention is important when you choose the manual tool change option or the gang tooling option. All tools mounted for use on the operator side of the workpiece are touched off using negativepositive X (diameter) values, and most X words in part programming for these tools have negativepositive values.

Set Work Offsets

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

E X A M P L E
If the active tool is 2 in. away from the spindle centerline, **4.000** displays in the **X** DRO field.

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

- Select **Zero**.

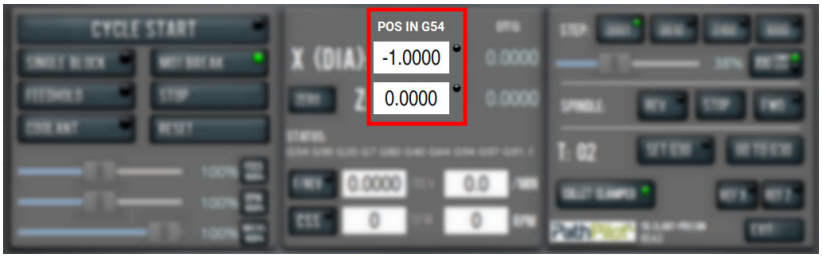


Figure 65: Work Offset DRO fields.

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.

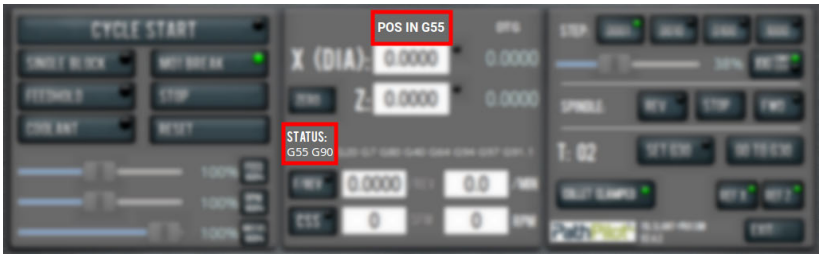


Figure 66: Work offset indicated in the PathPilot interface.

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" (on the next page).

About Work Offsets

Work offsets allow you to think in terms of X and Z coordinates with respect to the part, rather than thinking of them with respect to the machine position. This means that you can jog the machine to an arbitrary location (like the end of a workpiece) and call that location zero.
You can save up to 500 work offsets in PathPilot. The naming structure varies based on the offset number, as detailed in the following table.

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

View Work Offsets

To view the current work offset:

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

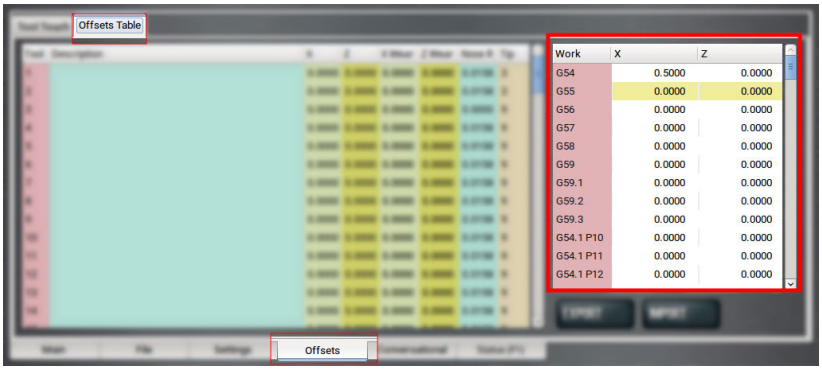


Figure 67: Work Offsets Table window.

The active work offset is highlighted.

To change the current work offset, go to "Set Work Offsets" (page 96).

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.

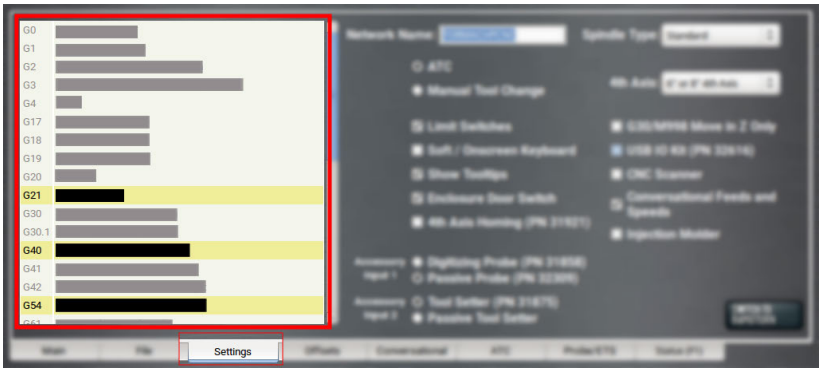


Figure 68: G-code Description window on the Settings tab.

RUN G-CODE PROGRAMS

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

Bring the Machine Out of Reset

➢ Select **Reset**.

Figure 69: Reset button.

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

About Reset Mode

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

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

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

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.

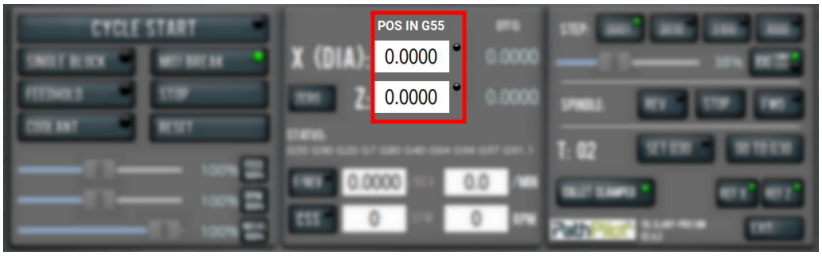


Figure 70: Work Offset DRO fields.

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

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 71: 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 72: Jog Speed slider.

For more information on continuous velocity mode, see "About Continuous Velocity Jogging" (on the next page).
To use step mode, select the step size. Do one of the following, depending on your accessories:

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

Figure 73: 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" (on the next page).

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 74: 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 75: Jog Speed slider.

About Continuous Velocity Jogging

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

- A keyboard key is pressed
- The Jog Shuttle outer ring is twisted away from the neutral position

This is useful when you're doing things like:

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

Jog in Step Mode

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

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

To select the step size:

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

Figure 76: Step buttons (in G20 mode).

About Step Jogging

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

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

Step jogging mode is useful to finely move the machine, like when you're indicating a workpiece or manually setting tool lengths.
The jog keys on the keyboard only move the machine in steps when step mode is indicated in PathPilot. The inner wheel on the jog shuttle always moves the machine in steps, regardless of which mode is indicated in PathPilot.

View the Current Machine Position

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

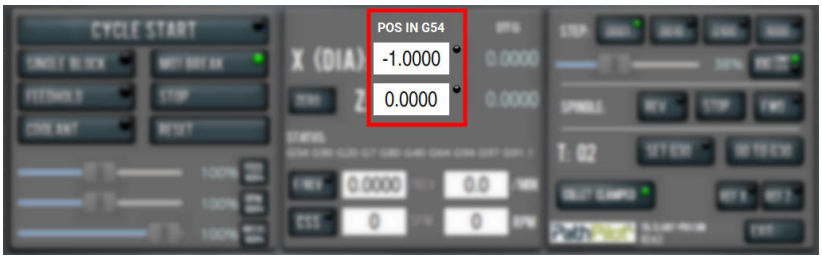


Figure 77: Work Offset DRO fields.

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 96).

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 78: 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).

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.

Start a Program

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

Figure 79: 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" (on the next page).

About Cycle Start

While a program is running, the Cycle Start button's light is on. Selecting Cycle Start causes an alarm unless the chuck guard or enclosure door is closed and a valid G-code program is loaded.
The Cycle Start button's light flashes if motion is paused during the program. The following modes may pause motion during a program:

- Single block
- Feed hold
- M01 break

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

Cycle Start Reference

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

- While you're not in the Main tab. For information, see "Main Tab" (page 19).
- Before you've loaded a G-code program. For information, see "Load G-Code" (page 35).
- Before referencing the machine. For information, see "Reference the Machine" (page 103).

Stop Machine Motion

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

Figure 80: Stop button.

Operate the Coolant Pump

To turn coolant on or off:

➢ Select **Coolant**.

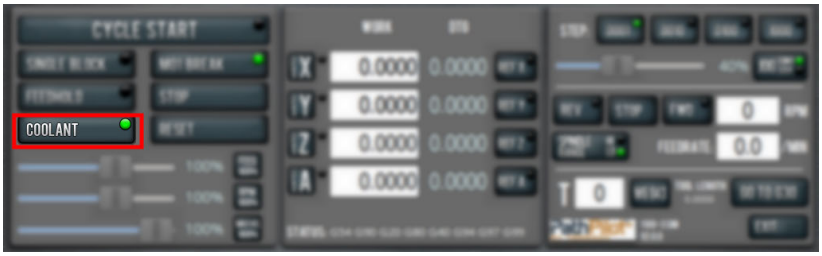


Figure 81: Coolant button.

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

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.

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.

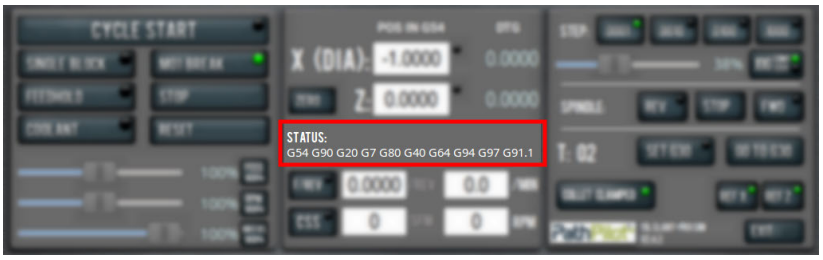


Figure 82: Status read-only DRO field.

For more information on G-code modes, go to "View Available G-Code Modes" (page 98).

View the Distance to Go

To view the distance to go:

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

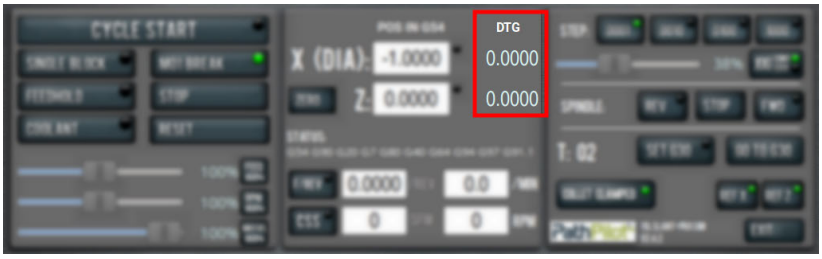


Figure 83: DTG read-only DRO fields.

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.

CONTROL G-CODE PROGRAMS

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

Use the Feed Hold Function

➢ Select **Feed Hold**.

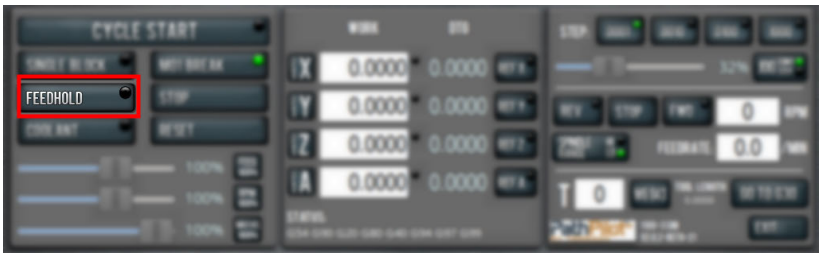


Figure 84: Feed Hold button.

> **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 104).

> **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.

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 85: 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 overriden).

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.

Use M01 Break Mode

➢ Select **M01 Break**.

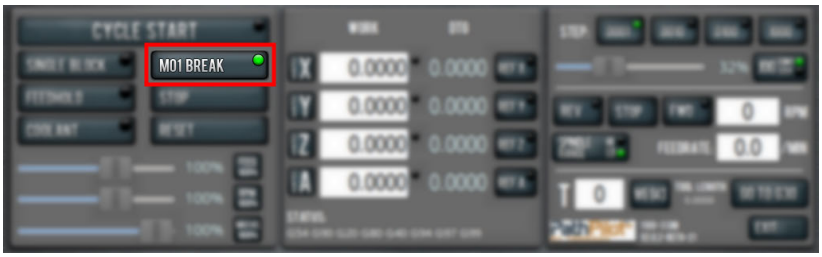


Figure 86: M01 Break button.

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 104).
- **When M01 Break is Inactive** PathPilot ignores all programmed M01 commands.

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 87: 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" (on the next page).

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%.

Use Single Block Mode

➢ Select **Single Block**.

Figure 88: 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 104).
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.

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 89: 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 overriden).

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%.

Change the Feed Rate

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

Figure 90: Feed Rate DRO field.

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

About Feed Rates

There are two feed rate modes:

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

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

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

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

Change the Spindle Speed

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

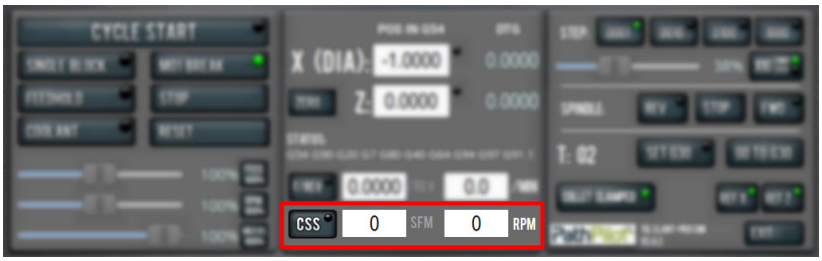


Figure 91: Spindle RPM DRO field.

For information, see "About Spindle Controls" (page 117).

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

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

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

There are two spindle speed command modes:

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

> **EXAMPLE**
> `G97 S500 M03` starts the spindle at 500 rpm.

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

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

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

Change the Feed Rate Mode

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

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

Figure 92: A value in the F/Rev DRO field, which turns on F/Rev mode.

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

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

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

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

Figure 93: A value in the F/Min DRO field, which turns off F/Rev mode.

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

- Select **F/Rev** when its light is off.

- Select the **MDI Line** DRO field and type `G94`. Then select the **Enter** key.
- Run a G-code program that has a G94 command.

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

About Feed Rate Modes

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

Units per Minute Mode (G94)

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

Units per Revolution Mode (G95)

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

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

Change the Spindle Speed Command Mode

To turn on constant surface speed mode (G96):

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

Figure 94: SFM DRO field.

To turn on constant RPM mode (G97):

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

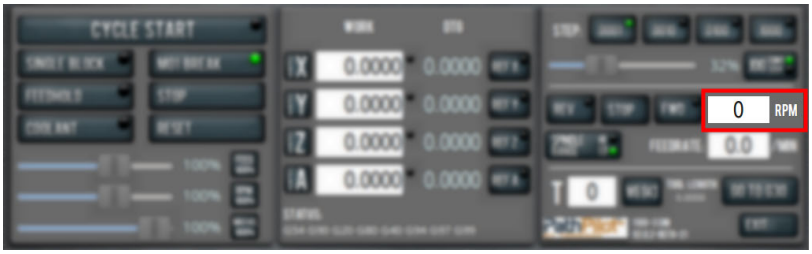


Figure 95: RPM DRO field.

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

About Spindle Speed Command Modes

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

Constant Surface Speed Mode (G96)

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

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

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

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

Constant RPM Mode (G97)

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

EXAMPLE
`G97 S500 M03` starts the spindle at 500 RPMs

About Spindle Controls

A spindle speed is the rate at which the spindle rotates.
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 RPM DRO field.
REV	M04	In G97 mode: start the spindle in the reverse direction at the RPM specified in the spindle RPM DRO field. In G96 mode: start the spindle at an RPM specified by the SFM DRO field and the X (Dia.) value in the X DRO field.
Stop	M05	Stop the spindle.

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

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

There are two spindle speed command modes:

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

  EXAMPLE
  `G97 S500 M03` starts the spindle at 500 rpm.

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

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

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

Change the Tool Number

The Tool DRO field shows the tool currently in the spindle.

Figure 96: Tool DRO field.

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

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

Figure 97: Tool DRO field.

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

Use a G30 Position

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

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

Figure 98: Set G30 button.

To go to a set G30 position:

- Do one of the following:
  - Use a `G30` command in a G-code program.
  - Select **Go To G30**.

Figure 99: 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 84).

> **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.

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.

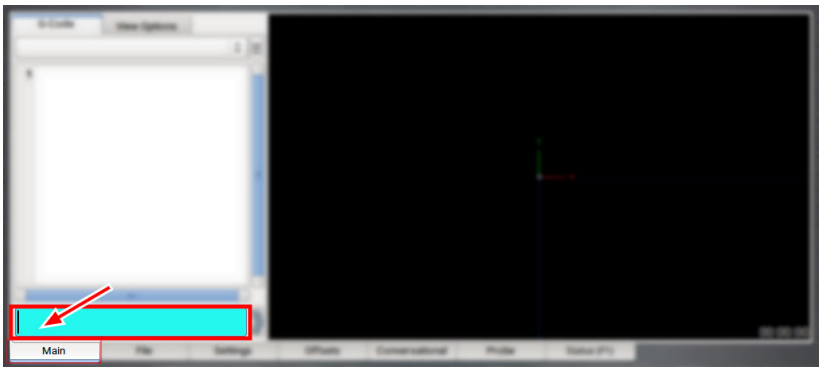


Figure 100: MDI Line DRO field.

   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 CALC	Open the calculator.
ADMIN CONFIG	Change the configuration of the PathPilot interface.
ADMIN DATE	Customize the PathPilot controller's date and time.
ADMIN DISPLAY	Customize the PathPilot controller's screen display.
ADMIN DROPBOX	Connect your controller to a Dropbox account.
ADMIN KEYBOARD	Customize the PathPilot controller's keyboard layout.
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 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 TOOLTIP DELAYMS	Set the milliseconds prior to displaying the tooltip (and then again for the expanded tooltip). The default is 1200 milliseconds.
ADMIN TOOLTIP MAXDISPLAYSEC	Limit the amount of time the expanded tooltip displays. The default is 15 seconds.
ADMIN TOUCHSCREEN	Configure the optional Touch Screen Kit.

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).

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 125).

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**.

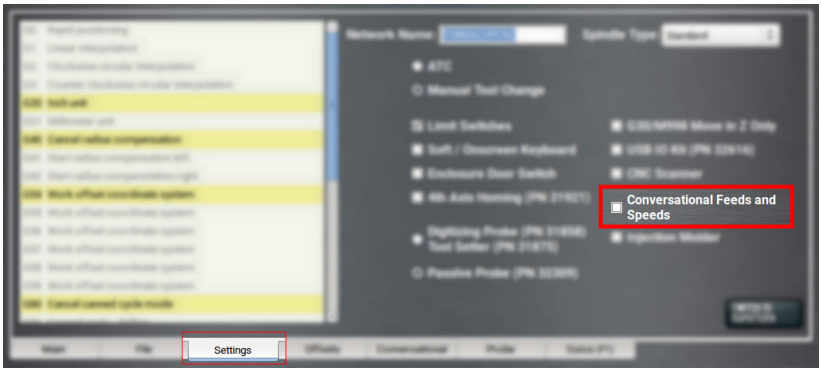


Figure 101: Settings tab.

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

Figure 102: Feeds and speeds suggestions on the Conversational tab.

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

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

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

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

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

Refresh DRO Field Values

The suggested values are no longer valid if:

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

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

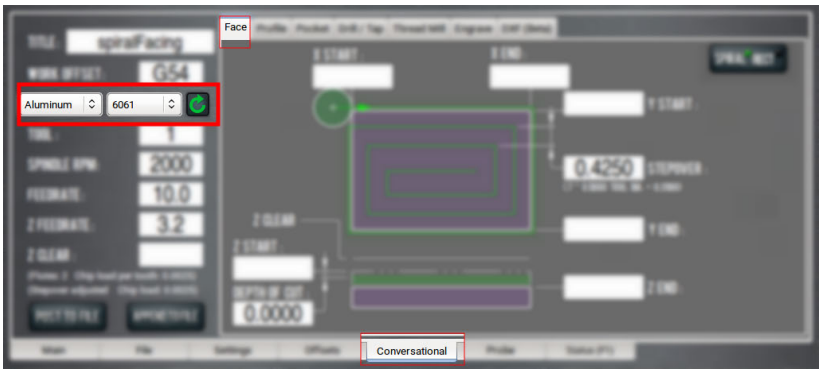


Figure 103: Refresh button on the Conversational tab.

Enable Feeds and Speeds Suggestions in Conversational Routines

You can use PathPilot to automatically calculate feeds and speeds. For more information, see "Use Feeds and Speeds Suggestions" (page 122).

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

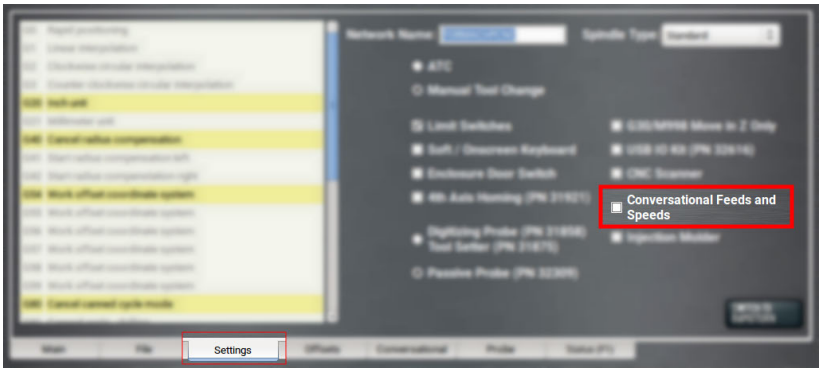


Figure 104: Settings tab.

Create Tool Descriptions

If desired, you can create tool descriptions in PathPilot. Detailed tool descriptions allow you to receive feeds and speeds suggestions in conversational programming. For information, see "Use Feeds and Speeds Suggestions" (page 122).

Manually Enter Tool Descriptions

PathPilot uses keywords and patterns in the tool description to recognize tooling features. For information, see"Tool Keywords Reference" (page 127).
To manually enter tool descriptions:

1. From the PathPilot interface, on the **Offsets** tab, select the **Offsets Table** tab, and identify the **Tool Table** window.
2. Select a blank line.

3. Type a description for the tool. Descriptions are not case sensitive.
   If a pattern or word in the description is recognized, PathPilot uses syntax highlighting to indicate a valid description.

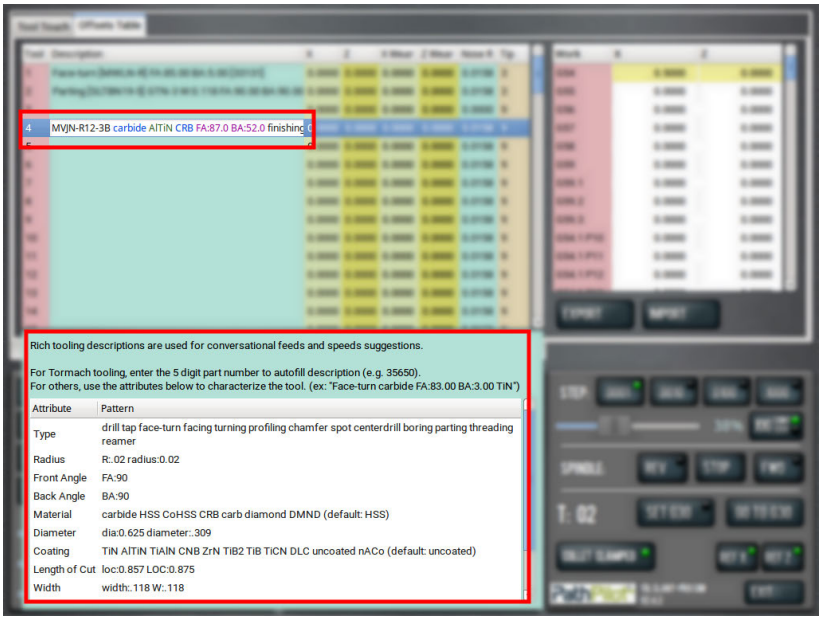


Figure 105: A manually-entered tool description.

Examples

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

> **E X A M P L E**
> MVJN-R12-3B carbide AlTiN CRB FA:87.0 BA:52.0 finishing

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

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

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

> **E X A M P L E**
> Gold colored tool bit from middle drawer

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

Automatically Generate Tool Descriptions

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

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

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

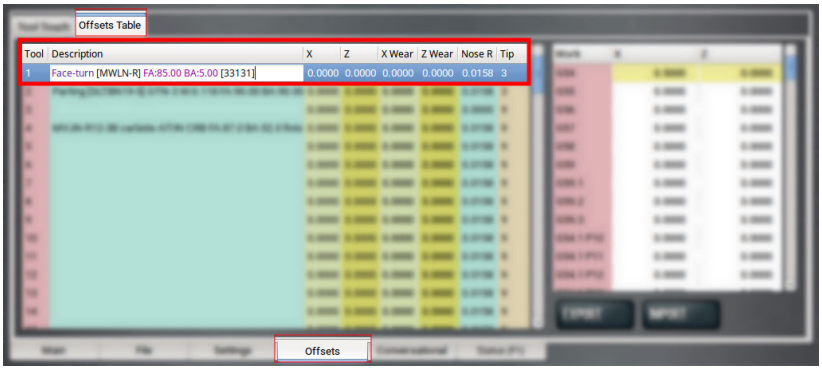


Figure 106: An automatically-generated tool description for a Tormach tool.

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

Tool Keywords Reference

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

Item	Pattern	Example	Notes
type	drill, tap, face-turn, facing, turning, profiling, chamfer, spot, centerdrill, boring, parting, threading, reamer	`DRILL`, `CHAMFER`, `BORING`	
tool radius	“R” or “radius”, followed by a colon, followed by a decimal number	`R:.0158`, `RADIUS:0.32`	No radius specified is the same as a zero radius.
front angle	“FA”, followed by a colon, followed by a decimal number	`FA:90`	
back angle	“BA”, followed by a colon, followed by a decimal number	`BA:90`	

Item	Pattern	Example	Notes
tool material	carbide, HSS, CoHSS, CRB, carb, diamond, DMND	`HSS`, `COHSS`, `CRB`	HSS: High-speed steel; CoHSS: Cobalt high-speed steel; CRB: Carbide; No tool material specified is the same as HSS.
tool diameter	“dia” or “diameter”, followed by a colon, followed by a decimal number	`DIA:.0341`, `DIAMETER:.750`	For axial tools (drills taps, reamers, etc.).
tool coating	TiN, AlTiN, TiAlN, CNB, ZrN, TiB2, TiB, TiCN, DLC, uncoated, nACo	`TIN`, `ZRN`, `TIB2`	No coating specified is the same as uncoated.
length of cut	“loc”, followed by a colon, followed by a decimal number	`LOC:0.875`	For axial tools (drills, reamers, etc.). If no length of cut is specified, a length will be assumed based on cutter diameter.
tool width	“width” or “w”, followed by a colon, followed by a decimal number	`W:.118`, `WIDTH:.118`	For grooving and parting tools.
min bore	“min-bore”, followed by a colon, followed by a decimal number	`MIN-BORE:.522`	
flutes	A number, followed by “FL” or “FLUTE”	`4FL`, `12FL`, `2FLUTE`	No flutes specified is the same as 2 flutes.

Use Cycle Counters (M30 and M99)

On the Main tab, the Tool Path display shows M30 and M99 cycle counters. They're useful to count parts completed during unattended operation. For each M-code, there's an A and B counter. This provides more flexibility, because you can reset them to 0 independently.
For example, you could use M30 A to count parts each shift, and M30 B to count parts each week. The cycle counters persist across the controller's power cycles.

Monitor Cycle Counters

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

Change Cycle Counter Values

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

Cycle Counter	Parameter
M30 A	#5650

Cycle Counter	Parameter
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.

SYSTEM FILE MANAGEMENT

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

Manage System Files

Use the File tab to manage system files on the PathPilot controller. For information, see "About System Files" (below).
To manage system files:

- From the PathPilot interface, on the **File** tab, do any of the following from the **Controller Files** window:
  - Select a file, and then select **New Folder**, **Rename**, or **Delete**.
  - Select a file, and go to the **Options** menu. Then, select **Copy**, **Cut**, or **Paste**.

To navigate through the system files:

- Select **Back** or **Home**.

About System Files

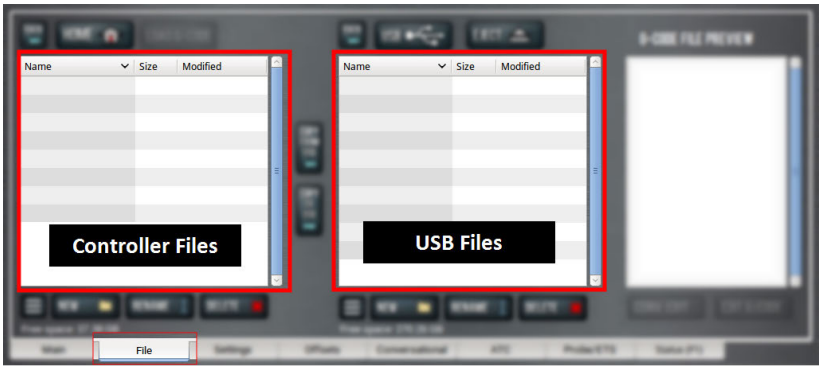


Figure 107: File tab.

PathPilot doesn't run G-code program files from a USB drive. You must first transfer files to the PathPilot controller. For information on transferring files, see "Transfer Files to and From the Controller" (page 35).

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.

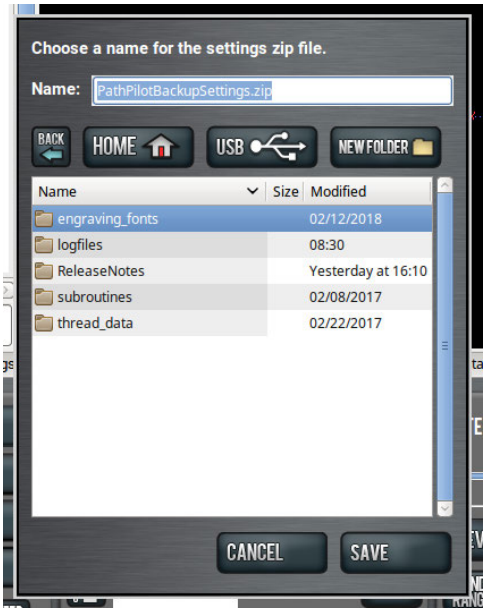


Figure 108: Admin Settings Backup dialog box.

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 109: Controller Files window on the File tab.

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

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

About Backup Files

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

Restore Backup Files

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

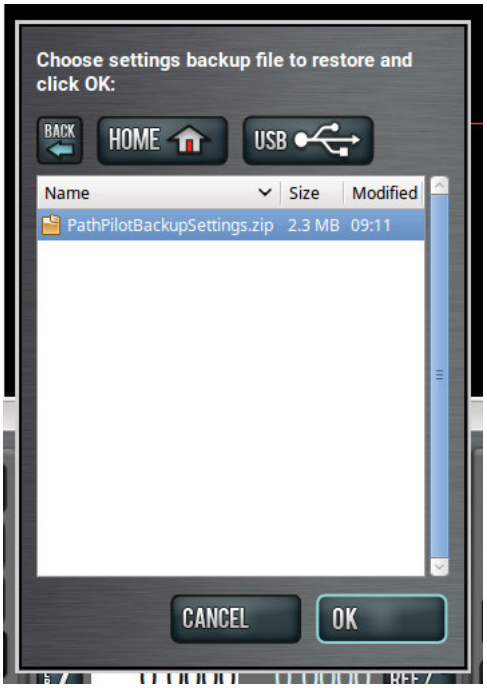


Figure 110: Admin Settings Restore dialog box.

3. From the **Admin Settings Restore** dialog box, navigate to the backup .zip file on the USB drive, and then select **OK**.
   The PathPilot operating system restores the backup, then restarts.

4. If you backed up any other files onto the USB drive, you must manually transfer the files to the PathPilot controller. From the PathPilot interface, on the **File** tab, in the **USB Files** window, select the files you want to transfer.

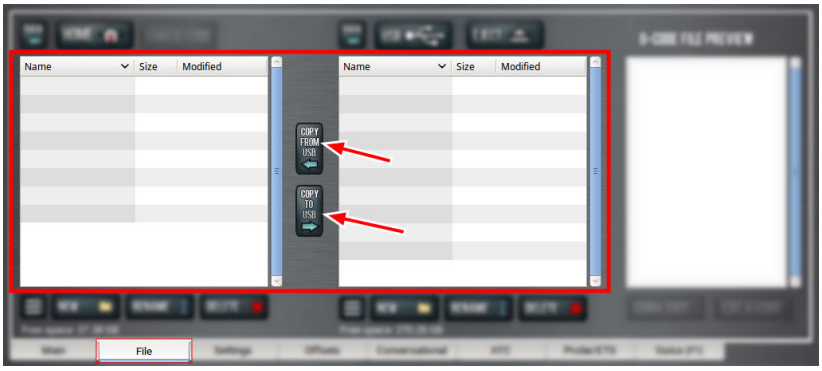


Figure 111: USB Files window on the File tab.

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

5. From the **Controller Files** window, select the folder into which you want to copy the files.
6. Select **Copy From USB**.
   The files display in the **Controller Files** window.

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

7. 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.

Import and Export the Tool Table

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

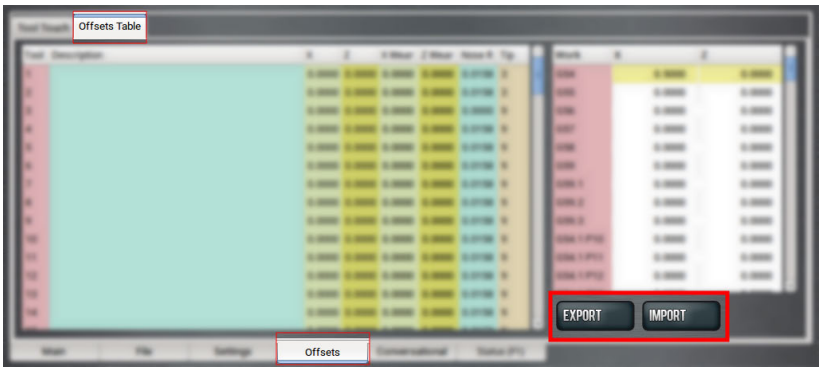


Figure 112: Export and Import buttons on the Offsets tab.

Import a .csv File

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

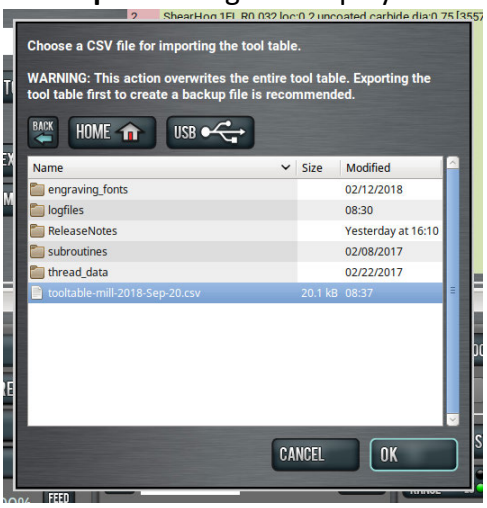


Figure 113: Import dialog box.

5. Navigate to the .csv file on the USB drive. Then, select **OK**.
   The .csv file updates the tool table.

Export the Tool Table as a .csv File

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

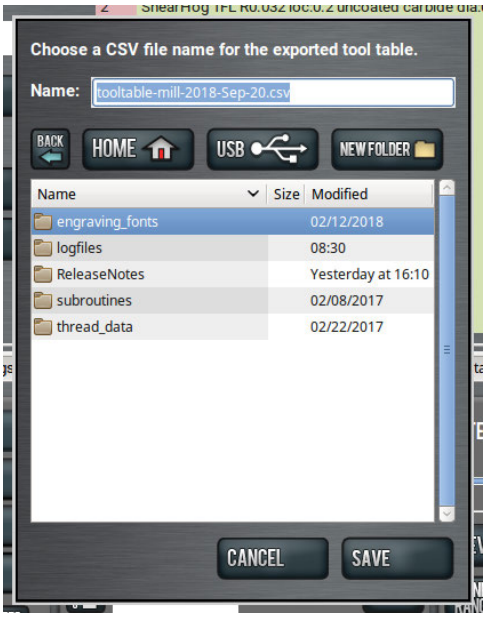


Figure 114: Export dialog box.

2. In the **Name** DRO field, type the name for the .csv file.
3. Select **Save**.
   The .csv file is saved in the **File** tab.
4. From the **File** tab, select the newly created .csv file, and then select **Copy to USB**.
5. Select **Eject**.
   It's safe to remove the USB drive from the controller.


TITLE: externalThread-Mill
Aluminum
-any-
WORK OFFSET: G59
TOOL: 2
SPINDLE RPM: 555
FEEDRATE: 100.0
Z FEEDRATE: 100.0
Z CLEAR: 0.6000
(Flutes: N/A Chip load per tooth: N/A)
(SFM: 0 – MRR: N/A)
POST TO FILE
APPEND TO FILE
Face
Profile
Pocket
Drill / Tap
Thread Mill
Engrave
DXF (Mill)
EXTERNAL
INTERNAL
DEPTH
X (FROM DRILL TABLE)
Y (FROM DRILL TABLE)
NUMBER OF PASSES:
THREADS/IN:
3.0
Z START:
3.0000
Z CLEAR
PITCH (IN):
0.3333
2.0000
Z END:
LEFT
RIGHT
Main
File
Settings
Offsets
Conversational
Probe
Status
CYCLE START
SINGLE BLOCK
M01 BREAK
FEEDHOLD
STOP
COOLANT
RESET
100%
FEED 100%
100%
RPM 100%
100%
MAXVEL 100%
POS IN G55
DTG
ZERO X
5.1931
0.0000
REF X
ZERO Y
0.6512
0.0000
REF Y
ZERO Z
0.0000
0.0000
REF Z
ZERO A
0.0000
0.0000
REF A
STATUS:
G55 G90 G20 G80 G40 G94 G97 G99 G91.1
STEP:
REV
SPINDLE RANGE
T
0
05:06:02

PROGRAMMING

IN THIS SECTION, YOU'LL LEARN:

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

CONTENTS

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.

Definitions

Read the following sections for reference:

Linear Axes

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

Controlled Point

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

There are nine possible orientations for the cutting point, identified by numbers shown in the following image (with orientation code 3 illustrated). If this tool is put in the turret, or a rear position on the gang plate, it's orientation 2.

Figure 1: Cutting point orientation graphic.

- Tools with **orientations 1, 2, 3, and 4** all turn to size and face. Their controlled points are the intersection of tangents to the cutting tip radius so the radius has no effect on the diameter or length of the part made. If, however, the cut is angled, then the radius means it does not cut the expected part but one a little larger than required.
  To machine a part corresponding to the angled tool path defined in the G-code, it is necessary to use tool-tip radius compensation (i.e., G41 or G42).
- Tools with **orientations 6 and 8** turn diameters to size and are often used for making profiles. It is difficult to accurately estimate their controlled point in the Z-direction.
- Tools with **orientations 5 and 7** turn on the face of stock and are unusual. It is difficult to accurately estimate their controlled point in the X-direction without a test cut.
- Tools with **orientation 9** are used when tool-tip compensation is applied in all moves.

Some orientations seem unusual at first glance, like a parting or grooving tool in the front tool post has orientation 4. This is because the program defines the part to the right of the tool. The size of the waste left in the chuck is unimportant.

Coordinated Linear Motion

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

Feed Rate

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

Arc Motion

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

Coolant

Although G-code provides for separate flood and mist coolant, either M07 or M08 turn on the lathe's coolant pump and M09 turns it off.

Dwell

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

Units

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

Current Position

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

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

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

Work Offsets

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

Selected Plane

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

Tool Table

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

Wear Offsets

Wear offsets are values used to fine tune a part program to compensate for things like tool wear or spring back. Wear offsets are applied with the T command as well, by specifying the desired wear offset register with the last two digits of a four-digit T command. For example, T02 applies the geometry offsets for tool 2, whereas T0202 applies both geometry offsets and wear offsets. Note that either command causes the turret to change position to pocket 2 if the machine is equipped with a turret. There is no M6 tool change command for lathes like there is for CNC milling machines.

Path Control Modes

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

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

Feed and Speed Override Controls

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

Block Delete Control

PathPilot doesn't implement the optional omission of blocks of code that are prefixed with the forward slash symbol (`/`).

Optional Program Stop Control

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

PROGRAMMING OVERVIEW

Read the following sections for a G-code overview:

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).

G-Code Formatting Reference

A permissible block of input code is made up of the following programming elements, in order, with the restriction that there is a maximum of 256 characters allowed on a line:

1. *(Optional)* Block delete character (`/`)
2. *(Optional)* Line number
3. Any number of words, parameter settings, and comments
4. End of line marker (carriage return or line break)

Programs are limited to 999,999 lines of code.
Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. Blank lines are allowed in the input, but they're ignored. Input is not case sensitive (except in

comments), so any letter outside a comment may be in uppercase or lowercase without changing the meaning of a line.

EXAMPLE
`G00 x +0. 12 34y 7` is equal to `G00 x+0.1234 y7`

A line may have:

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

Any input not explicitly allowed is illegal, and causes the interpreter to either signal an error or ignore the line. PathPilot omits blocks of code that are prefixed with a block delete character (`/`).
PathPilot sometimes ignores things it doesn't understand. If a command doesn't work as expected, or does nothing, make sure that it's correctly typed. PathPilot doesn't check for excessively high machining feeds or speeds, and it doesn't detect situations where a legal command will do something unfortunate (like machining a fixture).

Line Numbers

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

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

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

Words

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

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

Letters

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

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

Letter	Description
A	A-axis
B	B-axis

Letter	Description
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
X	X-axis
Y	Y-axis
Z	Z-axis

Values

A real value is one of the following:

- An explicit number (like 341, or -0.8807)
- An expression (like [2+2.4])

- A parameter value (like #88)
- A unary operation value (like acos[0])

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

A number is a subset of a real value. Processing a real value to come up with a number is called evaluating. An explicit number evaluates to itself.
Explicit numbers have the following rules (in this case, a digit is a single character, 0 through 9):

- A number must consist of the following, in the order listed:
  1. An optional plus or minus sign
  2. Zero to many digits
  3. *(Optional)* One decimal point
  4. Zero to many digits
- There must be at least one digit somewhere in the number.
- It must be either an integer (no decimal point) or a decimals (decimal point).
- It may have any number of digits (subject to line length limitations).

  > **Note:** PathPilot only keeps 17 significant figures, which is enough for all known applications.

- A non-zero number with no sign as the first character is assumed to be positive.

Initial zeros (a zero before the decimal point and the first non-zero digit) and trailing zeros (a zero after the decimal point and the last non-zero digit) are allowed, but not required. A number written with initial or trailing zeros has the same value when it is read as if the extra zeros were not there.
Numbers used for specific purposes by PathPilot are often restricted to some finite set of values, or to some range of values. In many uses, decimal numbers must be close enough to an integer to be accepted as a valid input. A decimal number which is supposed to be close to an integer is considered close enough if it is within 0.0001 of an integer.

Order of Execution

If a parameter setting of the same parameter is repeated on a line (like `#3=15 #3=6`), only the last setting takes effect. It's illogical, but not illegal, to set the same parameter twice on the same line.
The order of items on a line doesn't determine the order of execution on the commands.
Three types of items' order may vary on a line (as given earlier in this section):

- **Word** May be reordered in any way without changing the meaning of the line.
- **Parameter Setting** If it's reordered, there is no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter takes effect.

  E X A M P L E
  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.

  E X A M P L E
  `G40 G01 #3=15 (foo) #4=-7.0` has five items and means exactly the same thing in any of the 120 possible orders, like `#4=-7.0 G01 #3=15 G40 (foo)`, for the five items.

The order of execution of items on a line is critical to safe and effective machine operation. If items occur on the same line, they are executed in a particular order. To impose a different order (like to turn coolant off before the spindle is stopped), code the commands on separate blocks.
The order of execution is as follows:

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

   Note: This is not supported.

6. Change tool (`T`)
7. Spindle on/off (`M03`, `M04`, `M05`)
8. Save State (`M70`, `M73`, restore state (`M72`), invalidate state (`M71`)
9. Coolant on/off (`M07`, `M08`, `M09`)
10. Enable/disable overrides (`M48`, `M49`, `M50`, `M51`, `M52`, `M53`)
11. Operator defined commands (`M101` to `M199`)
12. Dwell (`G04`)
13. Set active plane (`G17`, `G18`, `G19`)
14. Set length units (`G20`, `G21`)

15. Cutter radius compensation on/off (G40, G41, G42)
16. Tool table offset on/off (G43, G49)
17. Fixture table select (G54 through G58 and G59 P~)
18. Set path control mode (G61, G61.1, G64)
19. Set distance mode (G90, G91)
20. Set canned cycle return level mode (G98, G99)
21. Home, change coordinate system data (G10) or set offsets (G92, G94)
22. Perform motion (G00 to G03, G12, G13, G80 to G89 as modified by G53)
23. Stop (M00, M01, M02, M30, M60)

Modal Groups

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

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

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

Group	Commands	Group Description
Group 1	{G00, G01, G02, G03, G33, G38.x, G73, G76, G80, G81, G82, G84, G85, G86, G88, G89}	Motion (one always in effect)
Group 2	{G17, G18, G19, G17.1, G17.2, G17.3}	Plane selection
Group 3	{G90, G91}	Distance mode
Group 4	{G90.1, G91.1}	Arc distance mode

Group	Commands	Group Description
Group 5	{G93, G94}	Feed rate mode
Group 6	{G20, G21}	Length units
Group 7	{G40, G41, G42, G41.1, G42.1}	Cutter compensation
Group 8	{G43, G43.1, G49}	Tool length offset
Group 10	{G98, G99}	Return mode in canned cycles
Group 12	{G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3}	Select work offset coordinate system
Group 13	{G61, G61.1, G64}	Path control mode
Group 14	{G96, G97}	Spindle control mode
Group 15	{G07, G08}	Lathe diameter mode

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

Group	Commands	Group Description
Group 4	{M00, M01, M02, M30, M60}	Program stop and program end
Group 7	{M03, M04, M05}	Spindle control
Group 8	{M07, M08, M09}	Coolant control (special case: `M07` and `M08` may be active at the same time)
Group 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.

Supported G-Codes Reference

G-Code	Description
G00	Rapid linear motion
G01	Linear motion at feed rate
G02	Clockwise arc at feed rate
G03	Counterclockwise arc at feed rate
G04	Dwell
G07, G08	Diameter / radius mode **Note:** The 15L Slant-PRO lathe and the RapidTurn both use `G07` (X positions displayed in diameter values). `G08` is not used or supported in PathPilot.
G10 L1	Set tool table
G10 L2	Set coordinate system
G10 L10	Set tool table – calculated – workpiece
G10 L11	Set tool table – calculated – fixture
G10 L20	Set coordinate system
G17, G18, G19	Plane selection
G20/G21	Length units
G28	Return to predefined position
G28.1	Return to predefined position
G30	Return to predefined position
G33	Spindle synchronized motion (like threading)
G33.1	Rigid tapping
G40	Cancel cutter compensation
G41/G42	Cutter compensation (left/right)
G41.1, G42.1	Dynamic cutter compensation
G43	Apply tool length offset
G49	Cancel tool length compensation

G-Code	Description
G53	Absolute coordinates
G54-G59.3	Select work offset coordinate system
G61/G61.1	Set exact path control mode
G64	Set blended path control mode
G73	High-speed peck drill
G76	Multi-pass threading cycle
G80	Cancel canned cycles
G81	Drilling cycle
G82	Simple drilling cycle
G83	Peck drilling cycle
G85	Boring cycle
G86	Boring cycle
G88	Boring cycle
G89	Boring cycle
G90, G90.1	Arc distance mode
G91, G91.1	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92, etc.
G93, G94, G95	Feed rate mode
G96, G97	Spindle control mode
G98	Initial level return / R-point level after canned cycles

PROGRAMMING G-CODE

Read the following sections as a G-code reference:

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.

Rapid Linear Motion (G00)

For rapid linear motion, program: `G00 X~ Y~`

- `X~` is the X-axis coordinate
- `Y~` is the Y-axis coordinate

This produces coordinated linear motion to the destination point at the current traverse rate (or slower, if the machine won't go that fast). It's expected that cutting won't take place when a `G00` command is executing. The `G00` is optional if the current motion mode is G00.
Depending on where the tool is located, follow these two basic rules:

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

Conditions

The motion differs if:

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

Depending on where the tool is located, there are two basic rules to follow for safety: if the Z value represents a cutting move in the positive direction (i.e., out of a hole), the X-axis should be moved first. If the Z value represents a move in the positive direction, the X-axis should be executed last.

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

Linear Motion at Feed Rate (G01)

For linear motion at feed rate (for cutting or not), program: `G01 X~ Z~`

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

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

Conditions

The motion differs if:

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

Troubleshooting

It's an error if:

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

Arc at Feed Rate (G02 and G03)

A circular or helical arc is specified using either `G02` (clockwise arc) or `G03` (counterclockwise arc). The axis of the circle is normal to the XZ plane of the machine coordinate system. The direction is viewing from above the lathe.
The motion differs if cutter radius compensation is active.
Two formats are allowed for specifying an arc: the center format and the radius format. In both formats, the `G02` or `G03` is optional if it's the current motion mode.

Radius Format Arc

For a clockwise arc in radius format, program: `G02 X~ Z~ R~`
For a counterclockwise arc in radius format, program: `G03 X~ Z~ R~`

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

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

Troubleshooting

It's an error if:

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

Center Format Arc

For a clockwise arc in center format, program: `G02 X~ Z~ I~ K~`
For a counterclockwise arc in center format, program: `G03 X~ Z~ I~ K~`

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

It's an error if:

- X and Z are both omitted
  The axis words are all optional except that at least one of X and Z must be used.
- I and K are both omitted
  I and K are optional except that at least one of the two must be used.
- The end point of the arc is the same as the current point.

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

Lathe Diameter Mode (G07)

To enter the diameter mode for axis X on a lathe, program: `G07`
When in the diameter mode the X axis moves on a lathe will be half the distance to the center of the lathe. For example, X1.2 turns a part with diameter 2.4.

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)

To define an entry in the tool table, program: `G10 L1 P~ X~ Y~ R~ I~ J~ Q~`

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

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

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

Troubleshooting

It's an error if:

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

Set Tool Table (G10 L10)

To change the tool table entry for tool P so that if the tool offset is reloaded with the machine in its current position and with the current G5x and G92 offsets active, program: `G10 L10 P~ X~ Z~ R~ I~ J~ Q~`

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

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

Troubleshooting

It's an error if:

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

Set Tool Table (G10 L11)

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

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

Troubleshooting

It's an error if:

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

Set Coordinate System (G10 L20)

`G10 L20` is similar to `G10 L2`, except that instead of setting the offset/entry to the given value, it is set to a calculated value that makes the current coordinates become the given value.
Program: `G10 L20 P~ X~ Z~`

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

Troubleshooting

It's an error if:

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

Plane Selection (G17, G18, G19)

To select the XY-plane as active, program: `G17`
To select the XZ-plane as active, program: `G18`

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

To select the YZ-plane as active, program: `G19`
The active plane determines how the tool path of an arc (`G02` or `G03`) or canned cycle (`G73`, `G81` through `G89`) is interpreted.

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.

Return to Predefined Position (G28 and G28.1)

To make a rapid linear move from the current position to the absolute position of the values in parameters 5161-5166: `G28`
To make a rapid linear move to the G28.1 position by first going to the intermediate position specified by the X~, and Z~ words, program: `G28 X~ Z~`

> **Note:** Any axis not specified won't move.

To store the current location of the tool in the G28.1 setting, program: `G28.1`
G28 uses the values stored in parameters 5161 and 5163 as the X and Z final point to move to. The parameter values are absolute machine coordinates in the native machine units of inches.
To store the current absolute position into parameters 5161-5163, program: `G28.1`

Troubleshooting

It's an error if:

- Cutter Compensation is turned on

Return to Predefined Position (G30 and G30.1)

G30 uses the values stored in parameters 5181 and 5183 as the X and Z final point to move to. The parameter values are absolute machine coordinates in the native machine units of inches.
To make a rapid traverse move from the current position to the absolute position of the values in parameters, program: `G30`
To make a rapid traverse move to the position specified by axes including any offsets, then make a rapid traverse move to the absolute position of the values in parameters 5181 and/or 5183, program: `G30 X~ Y~ Z~`

> **Note:** Any axis not specified won't move.

To store the current absolute position into parameters 5181-5183, program: `G30.1`

Troubleshooting

It's an error if:

- Cutter Compensation is turned on

Spindle-Synchronized Motion (G33)

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

- `K~` is distance per revolution

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

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

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

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

> **EXAMPLE**

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

Troubleshooting

It's an error if:

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

Rigid Tapping (G33.1)

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

- `K~` is distance per revolution

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

A rigid tapping move consists of the following sequence.

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

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

EXAMPLE

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

Troubleshooting

It's an error if:

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

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

Dynamic Cutter Compensation (G41.1 and G42.1)

To program dynamic Cutter Compensation to the left of the programmed tool path, program: `G41.1 D~ L~`
To program dynamic Cutter Compensation to the right of the programmed tool path, program: `G42.1 D~ L~`

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

`G41.1` and `G42.1` function the same as `G41` and `G42`, with the added scope of being able to ignore the tool table and to program the tool diameter.

Troubleshooting

It's an error if:

- Cutter Compensation is commanded to turn on when it is already on
- The L word isn't in the range from 0 to 9 inclusive

Absolute Coordinates (G53)

For rapid linear motion to a point expressed in absolute coordinates, program: `G01 G53 X~ Z~` (or use with `G00` instead of `G01`)
All the axis words are optional, except that at least one must be used. The `G00` or `G01` is optional if it is in the current motion mode. `G53` isn't modal, and must be programmed on each line on which it is intended to be active. This produces coordinated linear motion to the programmed point. If `G01` is active, the speed of motion is the current feed rate (or slower if the machine won't go that fast). If `G00` is active, the speed of motion is the current traverse rate (or slower if the machine won't go that fast).

Troubleshooting

It's an error if:

- `G53` is used without `G00` or `G01` being active
- `G53` is used while cutter radius compensation is on

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 97).

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).

Set Exact Path Control Mode (G61)

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

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.

Threading Cycle (G76)

To cut a thread in multiple passes, program: `G76 P~ Z~ I~ J~ R~ K~ Q~ H~ E~ L`

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

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

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

Optional Settings

- `R~` is the depth degression. R1.0 selects constant depth on successive threading passes. R2.0, which is usual, selects constant area. Values between 1.0 and 2.0 select decreasing depth but increasing area. Values above 2.0 select decreasing area. Beware that unnecessarily high degression values causes a large number of passes to be used (degression = a descent by stages or steps).
- `Q~` is the compound slide angle is the angle (in degrees) describing to what extent successive passes should be offset along the drive line. This is used to cause one side of the tool to remove more material than the other. A positive Q value causes the leading edge of the tool to cut more heavily. Typical values for threads with a 60° angle are 29, 29.5 or 30.
- `H~` is the number of spring passes. Spring passes are additional passes at full thread depth used to allow for any tool or workpiece deflection during the main cuts. If no additional passes are desired (for example on a work-hardening material), program H0.

- `E~` specifies the distance along the drive line used for the entry/exit taper. The angle of the taper is so the last pass tapers to the thread crest over the distance specified with E.' E0.2' will give a taper for the first/last 0.2 length units along the thread. For a 45° entry/exit taper, program E the same as K.
- `L~` specifies which ends of the thread get the taper. Program L0 for no taper (the default), L1 for entry taper, L2 for exit taper, or L3 for both entry and exit tapers. Entry tapers pause at the drive line to synchronize with the index pulse then feed in to the beginning of the taper. No entry taper and the tool will rapid to the cut depth, then synchronize and begin the cut.

The tool should be moved to the initial X and Z positions prior to issuing the G76. The X position is the drive line and the Z position is the start of the threads.
The tool pauses briefly for synchronization before each threading pass, so a relief groove is required at the entry unless the beginning of the thread is past the end of the material or if an entry taper is used.
Unless using an exit taper, the exit move (traverse to original X) is not synchronized to the spindle speed. With a slow spindle, the exit move might take only a small fraction of a revolution. If the spindle speed is increased after several passes are complete, subsequent exit moves requires a larger portion of a revolution, resulting in a very heavy cut during the exit move. This can be avoided by providing a relief groove at the exit, or by not changing the spindle speed while threading.
The final position of the tool is at the end of the drive line. A safe Z move is needed with an internal thread to remove the tool from the hole.

Troubleshooting

It's an error if:

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

Tapping Cycle (G84)

The `G84` cycle is intended for tapping. This cycle rotates the spindle clockwise to tap a pre-drilled hole; when the bottom of the hole is reached, the spindle rotates in the reverse direction and exits the hole.
Program: `G84 X~ Y~ Z~ R~ P~ F~`

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

The G84 cycle is as follows:

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

This cycle uses a P word, where P specifies the number of seconds to dwell. The P word is optional – if it is not included, PathPilot calculates a dwell for you (half of a second per 1000 RPM).
Spindle speed must be commanded before calling a G84 cycle. Feed rate override is ignored during a tapping cycle. Feedhold is ignored until the return operation is executed. After the tapping operation is completed, either a G98 or G99 command controls the return height — G99 returns the tool to the R-plane; G98 returns the tool to the initial height.

EXAMPLE

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

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.

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.

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`).

E X A M P L E
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`

Feed Rate Mode (G93, G94, and G95)

To set the active feed rate mode to inverse time, program: `G93`
Inverse time is used to program simultaneous coordinated linear and coordinated rotary motion. In inverse time feed rate mode, an F word means the move should be completed in ***[1/F number]*** minutes.

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

When the inverse time feed rate mode is active, an F word must appear on every line which has a `G01`, `G02`, or `G03` motion, and an F word on a line that does not have `G01`, `G02`, or `G03` is ignored. Being in inverse time feed rate mode does not affect `G00` (rapid traverse) motions.
To set the active feed rate mode to units per minute mode, program: `G94`
In units per minute feed rate mode, an F word is interpreted to mean the controlled point should move at a certain number of inches per minute, or millimeters per minute, depending upon what length units are being used.
To set the active feed rate mode to units per revolution mode, program: `G95`
In units per revolution mode, an F word is interpreted to mean the controlled point should move a certain number of inches per revolution of the spindle, depending on what length units are being used. G95 is not suitable for threading, for threading use `G33` or `G76`.

Troubleshooting

It's an error if:

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

Spindle Control Mode (G96 and G97)

To set constant surface speed mode, program: `G96 D~ S~`

- `D~` is the maximum spindle RPM.
  This word is optional.
- `S~` is the surface speed.

  Note: If `G20` is the active mode, the value is interpreted as feet per minute. If `G21` is the active mode, the value is interpreted as meters per minute

  EXAMPLE
  `G96 D2500 S250` (set constant surface speed with a maximum RPM of 2500, and a surface speed of 250).

When using `G96` (the most common mode of machine operation), X0 in the current coordinate system (including offsets and tool lengths) must be the spindle axis.
To set RPM mode, program: `G97`

Troubleshooting

It's an error if:

- S is not specified with `G96`
- A feed move is specified in `G96` mode while the spindle is not turning

PROGRAMMING M-CODE

Read the following sections for reference:

Supported M-Codes Reference

M-Code	Description
M00	Program stop
M01	Optional program stop
M02	Program end
M03, M04	Rotate spindle clockwise/counterclockwise
M05	Stop spindle rotation
M07, M08	Coolant on
M09	All coolant off
M10, M11	Unclamp/clamp automatic collet closer
M30	Program end and rewind
M48	Enable speed and feed override
M49	Disable speed and feed override
M64	Activate output relays
M65	Deactivate output relays
M66	Wait on an input **Note:** `M64` through `M66` is only useful with a **USB M-Code I/O Interface Kit (PN 32616)**.
M98	Call subroutine

M-Code	Description
M99	Return from subroutine/repeat
M301, M302, M303	USB camera control

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 88).

Spindle Control (M03, M04, and M05)

To start the spindle turning clockwise at the currently programmed speed, program: `M03`
To start the spindle turning counterclockwise at the currently programmed speed, program: `M04`
The speed is programmed by the S word.
To stop the spindle from turning, program: `M05`
It's okay to use `M03` or `M04` if the spindle speed is set to 0; if this is done, the spindle won't start turning. If later the spindle speed is set above 0, the spindle starts turning. It is permitted to use `M03` or `M04` when the spindle is already turning, or to use `M05` when the spindle is already stopped.

Coolant Control (M07, M08, and M09)

To turn coolant on, program: `M07`
To turn flood coolant on, program: `M08`
To turn all coolant off, program: `M09`
It's always okay to use any of these commands, regardless of what coolant is on or off.

Automatic Collet Closer Control (M10 and M11)

To unclamp the collet closer, program: `M10`
To clamp the collet closer, program: `M11`

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.

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).

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.

Set Current Tool Number (M61)

To change the current tool number while in MDI or manual mode, program: `M61 Q~`

- `Q~` is the tool number

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

Troubleshooting

It's an error if:

- `Q~` is not 0 or greater

Call Subroutine (M98)

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

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

The program must contain a letter O line with the number of the P word of the call (for instance O1, O125, O777). This O line is a label which indicates the start of the subroutine. The O line, and the associated G-code, is normally placed at the end of the program with other subroutines following an M2, M30 or M99, so it is not reached directly by the flow of the program.
By using parameters, values, or incremental moves, a repeated subroutine can make several roughing cuts around a complex path or cut several identical objects from one piece of material.
Subroutine calls may be nested: a subroutine may contain a M98 call to another subroutine. As no conditional branching is permitted, it is not meaningful for subroutines to call themselves recursively.

Return from Subroutine (M99)

To return from a subroutine, program: `M99`
Execution continues after the M98 G-code which called the subroutine.

PROGRAMMING INPUT CODES

Read the following sections for reference:

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.

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

Change Tool Number (T)

It's your responsibility to make sure that the machine is in a safe place for changing tools (for example, by using `G30`). This allows optimization of motion which can save time. You can provide a pause for manual intervention with `M00` or `M01` before the tool change.

Troubleshooting

It's an error if:

- A negative T number is used
- A T number larger than 99 is used

To select a tool, program: `T~~`
The first number indicates the tool to be selected and the geometry offsets entry for it in the tool table. The second number, which is optional, indicates the line of the tool table giving the wear offsets to be applied.

EXAMPLES

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

If you have an (optional) turret, numbers T01 through T08 automatically actuates the turret to select the chosen tool. Numbers greater than T08 are assumed to be gang tools, and the program continues after applying the relevant offsets.
If you select Manual Tool Change, the machine stops and displays the required tool number while Cycle Start LED flashes. The program continues when you select clicking Cycle Start.

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 — i'ts difficult to debug, and difficult for another operator to understand. Modern CAM virtually eliminates the need for it.
Read the following sections for reference:

Parameters

Read the following sections for reference:

Parameters Reference

The RS274/NGC language supports parameters. Parameters are analogous to variables in other programming languages. PathPilot maintains an array of 10,320 numerical parameters. Many of them have specific uses. The parameters that are associated with fixtures are persistent over time. Other parameters are undefined when the operating system is loaded. The parameters are preserved when the interpreter is reset. Parameters 1 to 1000 can be used by the code of part-programs.
There are several types of parameters of different purpose and appearance. The only value type supported by parameters is floating-point; there are no string, Boolean or integer types in G-code like in other programming languages. However, logic expressions can be formulated with Boolean operators (AND, OR, XOR, and the comparison operators EQ, NE, GT, GE ,LT, LE), and the MOD, ROUND, FUP and FIX operators support integer arithmetic.

Parameter Syntax

There are three types of parameters, numbered, named local, and named global. The type of the parameter is defined by its syntax:

- Numbered - `#4711`
- Named local - `#<localvalue>`
- Named global - `#<_globalvalue>`

Parameter Scope

The scope of a parameter is either global or local within a subroutine. The scope of each parameter is inferred from its syntax. Subroutine parameters and named local parameters have local scope. Named global parameters and all numbered parameters starting from #31 are global in scope. RS274/NGC uses lexical scoping. In a subroutine, only the local parameters defined therein and any global parameters are visible. The local parameters of a calling procedure are not visible in a called procedure.

Behavior of Uninitialized Parameters

Uninitialized global parameters and unused subroutine parameters return the value zero when used in an expression. Uninitialized named parameters signal an error when used in an expression.

Parameter Mode

The mode of a parameter can either be read/write or read-only. Read/write parameters may be assigned values within an assignment statement. Read-only parameters cannot be assigned values. They may appear in expressions, but not on the left-hand side of an assignment statement.

Persistence and Volatility

Parameters can either be persistent or volatile. When the operating system is powered off, volatile parameters lose their values and are reset to zero. The values of persistent parameters are saved in a disc file and restored to their previous values when the operating system is powered on again. All parameters except numbered parameters in the current persistent range (5163 to 5390) are volatile.

Intended Use

Numbered parameters in the range #31-#5000, named global, and local parameters are available for general-purpose storage of floating-point values, like intermediate results, flags, etc., throughout program execution. They are read/write (can be assigned a value). Subroutine parameters, numbered parameters #1-#30, and system parameters are read-only and not available for general use. Subroutine parameters are used to hold the actual parameters passed to a subroutine. Numbered parameters in the range of #1-#30 are used to access offsets of coordinate systems. System parameters are used to determine the current running version and are read-only.

Numbered Parameters Reference

A numbered parameter is recognized by the pound symbol (`#`) followed by an integer between 1 and 5399. The parameter is referred to by this integer, and its value is whatever number is stored in the parameter. A value is stored in a parameter with the (`=`) operator.
Example: `#3 = 15` (set parameter 3 to 15)
A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line: `#3=6 G01 X#3` is interpreted, a straight move to a point where X = 15 occurs before the value of parameter 3 is set to 6.
The # symbol takes precedence over other operations. For example, `#1+2` means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, `#[1+2]` does mean the value found in parameter 3.

The # character may be repeated; for example `##2` means the value of parameter whose index is the (integer) value of parameter 2. PathPilot maintains a number of read-only parameters. Only parameters for the relevant axes are maintained: (X Y Z A) for mill and (X Z) for lathe. The remaining parameters for unused axes are undefined.

Read-Only Parameters

- 1-30: Subroutine local parameters of call arguments. These parameters are local to the subroutine. For further information, see Programming with Subroutines later in this chapter
- 31-5000: G-code operator parameters. These parameters are global in G-code file
- 5061-5070: Result of G38.2 probe (X Y Z A B C U V W)
- 5161-5169: G28 home for (X Y Z A B C U V W)
- 5181-5189: G30 home for (X Y Z A B C U V W)
- 5210: 1 if G92 offsets are active, 0 if not
- 5211-5219: G92 offset (X Y Z A B C U V W)
- 5220: Current coordinate system number 1-9 for G54 - G59.3
- 5221-5230: Coordinate System 1, G54 (X Y Z A B C U V W R) – R denotes XY rotation angle around Z-axis
- 5241-5250: Coordinate System 2, G55 (X Y Z A B C U V W R)
- 5261-5270: Coordinate System 3, G56 (X Y Z A B C U V W R)
- 5281-5290: Coordinate System 4, G57 (X Y Z A B C U V W R)
- 5301-5310: Coordinate System 5, G58 (X Y Z A B C U V W R)
- 5321-5330: Coordinate System 6, G59 (X Y Z A B C U V W R)
- 5341-5350: Coordinate System 7, G59.1 (X Y Z A B C U V W R)
- 5361-5370: Coordinate System 8, G59.2 (X Y Z A B C U V W R)
- 5381-5390: Coordinate System 9, G59.3 (X Y Z A B C U V W R)
- 5399: Result of M66 – check or wait for input
- 5400: Current tool number
- 5401-5409: Tool offset (X Y Z A B C U V W)
- 5410: Current tool diameter
- 5411: Current tool front angle
- 5412: Current tool back angle
- 5413: Current tool orientation
- 5420-5428: Current position including offsets in current program units (X Y Z A B C U V W)

Subroutine Parameters Reference

Subroutine parameters are specifically reserved for call arguments. By definition, these are parameters #1-#30 and are local to the subroutine.

Named Parameters Reference

Named parameters work like numbered parameters, but are easier to read and remember. All parameter names are converted to lowercase and have spaces and tabs removed. Named parameters must be enclosed with `< >` marks.

`#<named parameter here>` is a local named parameter. By default, a named parameter is local to the scope in which it is assigned.

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.

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

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.

E X A M P L E
`[2.0 / 3 * 1.5 - 5.5 / 11.0]` is equivalent to `[[[2.0 / 3] * 1.5] - [5.5 / 11.0]]`
which is equivalent to `[1.0 - 0.5]`
which is
`0.5`

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false, and any non-zero number is equivalent to logical true.

Functions Reference

The available functions are:

- `ATAN[Y]/[X]`: Four quadrant inverse tangent
- `ABS[arg]`: Absolute value
- `ACOS[arg]`: Inverse cosine
- `ASIN[arg]`: Inverse sine
- `COS[arg]`: Cosine
- `EXP[arg]`: e raised to the given power (ex)
- `FIX[arg]`: Round down to integer
- `FUP[arg]`: Round up to integer
- `ROUND[arg]`: Round to nearest integer
- `LN[arg]`: Base-e logarithm
- `SIN[arg]`: Sine
- `SQRT[arg]`: Square root

- `TAN[arg]`: Tangent
- `EXISTS[arg]`: Check named parameter

Subroutines

Subroutines are subprograms that are called from inside another program.
Read the following sections for reference:

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 theif 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, than the code following the else keyword is executed.

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

```
o102 endif
```

Repeating Subroutines Reference

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

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

Looping Subroutines Reference

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

do/while

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

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

while/endwhile

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

```
While/endwhile example:
(draw a sawtooth shape)
G00 X1 Y0 (move to start position)
#1 = 1 (assign parameter #1 the value of 0)
F25 (set a feed rate)
o101 while [#1 LT 10]
```

```
G01 X0
G01 Y[#1/10] X1
#1 = [#1+1] (increment the test counter)
o101 endwhile
M02 (end program)
```

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

- A return or endsub not within a sub definition
- A label on repeat which is defined elsewhere
- A label on while which is defined elsewhere and not referring to a do
- A label on if defined elsewhere
- A undefined label on else or elseif
- A label on else, elseif or endif not pointing to a matching if
- A label on break or continue which does not point to a matching while or do
- A label on endrepeat or endwhile no referring to a corresponding while or repeat


TITLE: externalThread-Mill
Aluminum
-any-
WORK OFFSET: G59
TOOL: 2
SPINDLE RPM: 555
FEEDRATE: 100.0
Z FEEDRATE: 100.0
Z CLEAR: 0.6000
(Flutes: N/A Chip load per tooth: N/A)
(SFM: 0 – MRR: N/A)
POST TO FILE
APPEND TO FILE
Face
Profile
Pocket
Drill / Tap
Thread Mill
Engrave
DXF (Mill)
EXTERNAL
INTERNAL
DEPTH
X (FROM DRILL TABLE)
Y (FROM DRILL TABLE)
NUMBER OF PASSES:
THREADS/IN:
3.0
Z START:
3.0000
Z CLEAR
PITCH (IN):
0.3333
2.0000
Z END:
LEFT
RIGHT
Main
File
Settings
Offsets
Conversational
Probe
Status
CYCLE START
SINGLE BLOCK
M01 BREAK
FEEDHOLD
STOP
COOLANT
RESET
100%
FEED 100%
100%
RPM 100%
100%
MAXVEL 100%
POS IN G55
DTG
ZERO X
5.1931
0.0000
REF X
ZERO Y
0.6512
0.0000
REF Y
ZERO Z
0.0000
0.0000
REF Z
ZERO A
0.0000
0.0000
REF A
STATUS:
G55 G90 G20 G80 G40 G94 G97 G99 G91.1
STEP:
REV
SPINDLE RANGE
T
0
05:06:02

PREVIOUS UPDATES

IN THIS SECTION, YOU'LL LEARN:

- About the enhancements and fixed issues in previous versions of PathPilot.

CONTENTS

RELEASE NOTES FOR PATHPILOT V2.9.6

May 2023

Enhancements

All

- We added support for Python-based diagnostic tests (.tsd files) to make some tech support steps quicker and easier. (PP-3698)

Mills

- We fixed an issue on MX mills where the Encoder self-test would, in rare cases, report false-positive failures during spindle acceleration. (PP-3917)

Plasma

- We added cut charts for Powermax 65, 85, and 105 SYNC plasma sources to the Auto FS material picker. (PP-3932)

Fixed Issues

Plasma

- We fixed an issue with set-start-line behavior on 1300PL plasmas, where arc voltage and feed settings were sometimes not updated at the start line. (PP-3476)

xsTECH

- We fixed an issue where PathPilot was prevented from functioning with some xsTECH routers. (PP-3938)

RELEASE NOTES FOR PATHPILOT V2.9.5

May 2023

Fixed Issues

- We fixed an issue introduced in PathPilot v2.9.4 where PathPilot was prevented from functioning with xsTECH routers. (PP-3914)
- We fixed an issue where some xsTECH routers E-stopped due to joint following errors on PathPilot 2.9.x. (PP-3667)

RELEASE NOTES FOR PATHPILOT V2.9.4

April 2023

Enhancements

- We added support for 7i92T machine interface boards, which are used with new PCNC 440 mills and 15L Slant-PRO lathes. (PP-3885)

RELEASE NOTES FOR PATHPILOT V2.9.3

February 2023

Enhancements

Lathes

- We restored the original default rapid speed on the 15L Slant-PRO lathe to 60 ipm because of infrequent, but ongoing, issues for some customers. (PP-3760)

Fixed Issues

All

- We fixed an issue where X/Y DROs were always in inches in rotated coordinate system. (PP-3774)
- Dropbox support now requires updating the Firefox browser on your controller. To easily update the browser, you can now use a script from the **Examples** folder in the **File** tab. (PP-3796)
  To learn how to use the technical support script, click here to read our how-to article.

RELEASE NOTES FOR PATHPILOT V2.9.2

October 2022

Enhancements

All

- Previously, loading a file larger than 2 MB disabled syntax highlighting and required you to manually enable it again. Now, syntax highlighting is restored to its previous setting when a file smaller than 2 MB is loaded. (PP-3675)

Fixed Issues

All

We fixed issues where:

- The previous tool live plot didn't clear when set start line was used with **Show Only Current Tool**. (PP-3595)
- The **Show Only Current Tool** setting sometimes reset after editing and reloading a G-code file. (PP-3596)
- In rare situations, a lot of continuous user interaction (like loading programs or conversational code/edits) resulted in low system resources and poor performance. (PP-3630)

Lathes

We fixed issues where:

- Conversational edits sometimes asked you to save changes again even though they had already been saved. (PP-3417)
- The minimum "X" extent of the tool path displayed incorrectly in the preview. (PP-3658)

Router

- We fixed an issue where, during occasional automated tool changes, a “bad number format” error appeared on the **Status** tab. (PP-3685)

RELEASE NOTES FOR PATHPILOT V2.9.1

August 2022

Fixed Issues

Mills

- We fixed an issue in PathPilot v2.9.0 where the **VFD Running** LED on the **Status** tab didn't illuminate on PCNC 440, PCNC 770, and PCNC 1100 mills. (PP-3632)

Lathes

- We fixed an issue in PathPilot v2.9.0 where, in G21 mode, some position and offset DROs displayed in the wrong units. (PP-3631)

RELEASE NOTES FOR PATHPILOT V2.9.0

July 2022

Enhancements

We added:

- An example spindle warmup program in the **Examples** folder in the **File** tab. Whenever the machine has been idle overnight or longer, running the spindle warmup program provides proper lubrication to the spindle bearings. For the 24R specifically, there's also a new spindle break-in program. (PP-3376)
- Driver support for new versions of USB WiFi adapters. (PP-3377)
- Detection of some error conditions in mills and lathes. A running program or MDI command now stops with an error if:
  - A motion requiring specific spindle rpm doesn't see the spindle reach the target speed within 20 seconds.
  - A motion requiring spindle synchronization doesn't see an index pulse within 10 seconds. (PP-3386)
- Support for the automatic tool changer (ATC) for 24R routers. (PP-3394)
- Support for the ZA6 robot. (PP-3395)
- Dropbox support now requires updating the Chrome browser on your controller. To easily update the browser, you can now use a script from the **Examples** folder in the **File** tab. (PP-3593)
  To learn how to use the technical support script, click here to read our how-to article.

Fixed Issues

All

We fixed issues where:

- In very rare situations, PathPilot unexpectedly shutdown while editing a conversational G-code program, though it could occur in other situations. (PP-3318)
- It wasn't possible to download files from PathPilot HUB if they were contained in a folder that didn't already exist on the local controller. (PP-3333)
- In some cases, the G71 path preview was shown incorrectly after a conversational edit session (program execution was not affected). (PP-3341)
- Introduced in PathPilot v2.8.2, a full-turn helical path could, in some cases, have too small of a pitch (due to an extra full turn). (PP-3354)
- An error could occur when downloading files from PathPilot HUB that are inside of folders. (PP-3390)
- Tool descriptions with colons were truncated during programs where manual tool changes were required. (PP-3431)

Mills

We fixed issues where:

- The maximum Z feed rate for conversational tapping was too small in G21, preventing you from saving an otherwise valid tapping program. (PP-3301)
- In some cases, G30 in conversational code failed with a limit error when the G30.1 Z position was at maximum Z height. (PP-3313)
- If G33.1 was called via MDI as the very first command after selecting **Ref** or **Reset**, it failed with an error, and caused the next MDI command to pause indefinitely before execution (which required you to select **Stop**/**Reset**). (PP-3328)
- A G-code error after a tool change in a running program could leave the previous tool's offset active. (PP-3357)
- On MX mills, calling a tool change reset the G28 position. (PP-3371)

Lathes

We fixed issues where:

- G71 conversational profile gouged if the profile crossed X0 with cutter compensation active. (PP-3277)
- On 8L lathes, warnings about incorrect tool offset X values for front/rear tool posts were reversed. (PP-3312)
- Ring-jogging in Y would eventually cause a soft emergency stop due to a joint following error on the non-existent Y axis. (PP-3334)
- Introduced in PathPilot v2.8.3, the G30 X position was incorrectly scaled. (PP-3344)
- Some rear tool post only tools were visible in the tool selection screen as possible tool types for the 8L lathe, which only uses front tool post tools. (PP-3369)
- In some cases, invalid tool numbers greater than 9999 were accepted in G-code, leading to spurious python exceptions in log files during conversational operations. (PP-3370)
- An explicit tool change to T0 didn't completely clear the tool offset if a wear offset was previously active (subsequent tool changes/offsets were unaffected). (PP-3378)
- In very rare situations, attempting to reference the X- and Z-axis at the same time on the 15L Slant-PRO lathe could result in one of the axes moving in the opposite direction it normally travels to reference. (PP-3383)
- In conversational profiling, front tool post tools weren't correctly configured and displayed. In some cases, the default front/back angles were incorrect, requiring an override to get proper profiling behavior. Also, we fixed the display of front tool post tool shapes in conversational profiling so that they match the lathe's orientation. (PP-3400)
- Recommended feed/speed/thread passes weren't populated for some materials on the 8L lathe. (PP-3427)

Plasma

We fixed issues where:

- In some cases, using set start line failed with a soft limit error if the start line was after a G15. (PP-3458)
- Unexpected AXIS_ABORT error messages sometimes appeared on the **Status** tab after a change in G20/G21 units during MDI actions or while running programs. (PP-3435)

RELEASE NOTES FOR PATHPILOT V2.8.3

September 2021

Enhancements

Mills and Routers

- For some setups (depending on the machine configuration and the model of the tool setter), we reduced the Z traverse feed rate in the G37 / G37.1 tool touchoff sequence. This reduces the risk of damage due to an incorrectly set G37 Z position. (PP-3291)

Mills

- We improved collision avoidance when using long tools in Automatic Tool Changers (ATCs). Previously, after a tool change, the final Z position was set to whichever was higher between G30 Z and tool change Z. Now, the Z position is saved at the beginning of a tool change, and later restored after the tool change is complete (if it's above the tool change height position). (PP-3286)

  Note: To toggle this behavior on or off, use the following commands in the MDI Line DRO field:
  `ADMIN ATC M6_EPILOG_RETRACT ON`
  `ADMIN ATC M6_EPILOG_RETRACT OFF`

xsTECH

- The noise made by the axis motors on xsTECH routers is now quieter. (PP-3256)

Fixed Issues

All

- We added updated graphics drivers for improved OpenGL performance on newer PathPilot controllers (Rev C). (PP-3279)
- We fixed an issue with very small G02 and G03 arcs and cutter compensation where, in some situations, the compensated arc used the wrong number of full turns. (PP-3278)
- The jog mode button's LED now reflects the current keyboard jogging mode. Previously, using the jog shuttle or console pendant could have side effects such that selecting the jog mode button appeared to have no effect in toggling the LED. (PP-3300)

Mills

- We fixed an issue with M98 subroutines/looping where the loop count was intermittently wrong after a program abort. (PP-1969)

- G30 now accepts an intermediate axis position in incremental (G91) coordinates. This makes it easier to retract individual axes. Previously, intermediate Z moves had to be to an absolute position. (PP-3273)

  EXAMPLE

  ```
  G91 G30 Z0
  G90
  ```

  Code breakdown: Move incrementally zero units in Z (no motion), then rapid in Z to the position specified in parameter 5183 (no other axes move).

Lathes

- We fixed an issue where, in some situations while using a console, cycling through jog axes from X to Z on the pendant reported a false error message that the maximum jog speed for the (non-existent) Y-axis was missing. (PP-3288)

  Note: This issue didn't affect keyboard jogging on the console.

RELEASE NOTES FOR PATHPILOT V2.8.0

June 2021

Enhancements

All

- We improved the behavior of the set start line function in two ways (PP-3090):
  1. If the correct tool is already in the spindle, skip the retract to tool change height.
  2. Retract to the `G30` position rather than `G53 Z0`. This saves time and allows for better control of retract position, like on lathes with a turret.
- We added a new, read-only parameter `# <_tool_offset_number>` to get the currently applied tool offset number at the interpreter read-ahead location for use in G-code programs. (PP-3249)

Mills

- We improved collision avoidance when using long tools in Automatic Tool Changers (ATCs). Now, after a tool change involving the ATC, the final Z position is set to whichever is higher between G30 Z and tool change Z. (PP-3254)

Plasma

- We added mild steel cut charts for the Powermax 85 plasma source. (PP-2941)
- We added a parametric conversational shape library. It includes things like common shapes, flanges, and brackets — along with conversational parameters — for quick and easy cutting. (PP-3193)

Fixed Issues

All

- We fixed issues where:
  - While using the pendant on the PathPilot operator console, the jog speed wasn't limited before the axes were referenced. (PP-3037)
  - The View Options tab (in the G-Code window) wouldn't refresh when a file modification was detected. (PP-3102)
  - Some G-code program errors would appear to leave the Tool Path display blank. (PP-3114)
  - When exiting a `G37` move, the machine moved to the initial Z position before moving to the Z clear position. (PP-3132)
  - While using the pendant on the PathPilot operator console, the axis motion stuttered when jogging at high speeds. (PP-3188)
  - After unplugging and re-connecting the touch screen monitor's USB cable, the calibration settings were removed. (PP-3245)

- Attempting to assign a value to a read-only parameter correctly raised an error, but failed to clear that error. This caused any future MDI or program runs to fail, and required a restart of PathPilot to resume normal operation. (PP-3252)

Mills

- We added rapid jogging support for rotary axes. (PP-2901)
- We fixed an issue where, if an ATC had low air pressure, selecting ATC FWD or ATC REV caused the PathPilot interface to freeze. This required a restart of PathPilot to resume normal operation. (PP-3253)

Lathes

- We fixed an issue where, after a spindle stop was commanded, a user command to unclamp the collet was allowed before the spindle had fully stopped. (PP-3219)

Plasma

- We fixed an issue where some AutoFS presets for mild steel used the incorrect amperage. (PP-3111)

RELEASE NOTES FOR PATHPILOT V2.7.4

May 2021

Enhancements

Mills

- On PCNC 440 Automatic Tool Changers (ATCs) that use the older, off-board stepper driver, we reduced the acceleration and deceleration parameters by 33% to dampen jerk during backlash compensation moves. (PP-3226)

Fixed Issues

All

- We fixed an issue where, while a program was running, you couldn't select the **Expand View** button in the **Notebook** section of the **Main** tab. This issue was introduced in PathPilot v2.7.3. (PP-3227)
- Previously, you could change the set-start-line options while a program was running, but the changes didn't take effect because they're cleared after program stop or completion. This has been resolved and now changes to set-start-line options aren't allowed while a program is running.

Mills

- We fixed issues where:
  - In some situations, using non-Tormach USB devices or hubs prevented the ATC firmware from updating. (PP-3221)
  - On PCNC and M series mills, some normal usages of the R / P word in G74 were incorrectly reported as an error. This issue was introduced in PathPilot v 2.5. (PP-3235)

RELEASE NOTES FOR PATHPILOT V2.7.3

April 2021

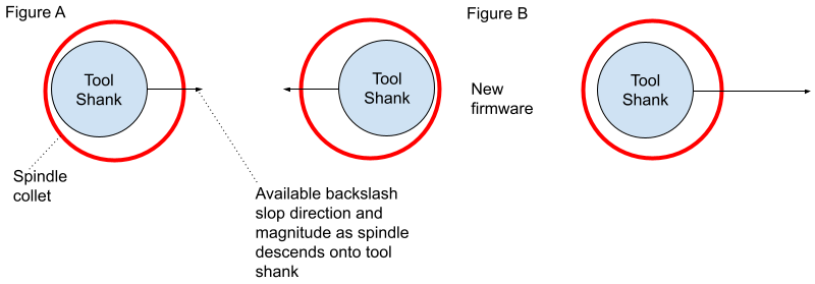
Enhancements

All

- We improved the diagnostic information that displays when the controller fails to power on because of a depleted CMOS battery. (PP-3173)
- While referencing an axis, you can now use the **Space Bar** key on the keyboard to use the feed hold function and stop axis motion. (PP-3178)
- We added driver support for newer Realtek 8152/8153 Ethernet chips, which provides wider support for USB to Ethernet adapters. (PP-3220)

Mills

- We updated the ATC firmware to make it easier to align the tool shank or BT30 pull stud with the spindle collet. Previously, adjusting the tray position required incremental rotations in both directions (see **Figure A**). Now, to compensate for any backlash in the tray position, the ATC always approaches the final target position from the same direction (see **Figure B**). For example, during forward moves, the ATC slightly overshoots the target position, and then reverses back into position.



> **IMPORTANT!** After you update the ATC firmware, we recommend that you check the adjustment of the tray position for a tool change. Before making any adjustments, keep the spindle collet above the tool shank or BT30 pull stud — when you're adjusting the target position using the **--** and **++** buttons on the **ATC** tab, the backlash compensation move still occurs.

If needed, the MDI command `ADMIN ATC BACKLASH OFF` disables the backlash compensation.

Fixed Issues

All

- We fixed issues where:
  - In conversational serial number engraving operations, G47 was placed after M30 when coolant was disabled. (PP-3156)
  - In conversational engraving operations, 'GG5x' was added if the text field was empty and the serial number box was checked. (PP-3162)
  - The preview in the **Tool Path** display wasn't updated if a program loaded with an error, which could be misleading. (PP-3163)
  - In simulation/PathPilot HUB configurations, limit switches were prevented from being disabled. This sometimes led to an unrecoverable state, like if the machine was jogged into a limit switch before all axes were referenced. (PP-3166)
  - A previously selected start line would sometimes be reset after running a long MDI command. (PP-3167)
  - During MDI command execution where keyboard jog commands would cause status messages and, in some cases, abort the MDI command in progress. (PP-3174)
  - Setting the G30 position to an axis G53 zero position sometimes caused G30 commands to generate limit errors that showed very small distance violations. (PP-3203)
  - The G30 position for the A-axis in G21 (metric) mode wouldn't position the axis correctly. (PP-3204)
  - Two display issues occurred with A-axis values in G21 mode:
    - A-axis work offsets were incorrectly scaled up by 25.4 in the **Work Offset** table (but correctly applied during program execution).
    - **Feed Rate** DRO field showed an incorrect value when only the A-axis was moving. (PP-3199)
- We improved the usability of the **Feed Hold** button (and its shortcut on the keyboard, the **Space Bar** key) in several conditions:
  - The **Cycle Start** button's LED now flashes when the machine is in feed hold during an MDI command
  - Feed hold causes a controlled stop of axis motion during jogging and referencing
  - We also fixed an issue where feed hold could be enabled during jogging, which prevented programs from running until explicitly cleared by a stop or selecting the **Reset** button. (PP-3178)

Mills

- We fixed issues where:
  - PCNC 440 configurations didn't support more than one USB I/O module. (PP-3157)
  - In certain situations, the conversational pocket operation could produce problematic tool paths. (PP-3172)

- When referencing PCNC 440 mills, the **Stop** and **Reset** buttons stopped the currently referencing axis, but didn't stop any other queued reference commands. (PP-3179)
- Set start line could, in some cases, report a false error when used with a program containing G37 with a non-zero P word. (PP-3195)
- Conversational Edit could silently fail to save a file if one or more operations were posted with Tool 0. (PP-3196)

Lathes

- We fixed a long-standing issue where, if a set start line was chosen after a G96 command, the maximum spindle speed (D word) would sometimes be ignored until the next explicit G96 line.
  We also improved lathe surface feed and RPM DRO fields so that they show the correct RPM-limited values during manual operations and jogging. Previously, the value in the RPM DRO field showed a value larger than the actual spindle speed for small X diameters. (PP-3165)
- We fixed an issue where the **Collet Clamped** button wasn't useable during M0/M01 stops. (PP-3217)

RELEASE NOTES FOR PATHPILOT V2.7.2

January 2021

Enhancements

All

- We improved the usability of toolpath extents display: the extents no longer include G28/G30 motions, or lead-in moves for set-start-line. (PP-3105)

Mills

- We optimized/reduced time spent waiting for the spindle to be at speed at the start of a G84 non-rigid tapped hole. (PP-3150)

Fixed Issues

All

- We fixed issues where:
  - The A-axis display selection wasn't enabled on start up, requiring you to explicitly disable and re-enable it. (PP-3119)
  - In some rare situations, or when setting up A-axis probing for the first time, the stock/axis center settings didn't match the setting shown in the interface. Also, A-axis probing now starts from the current A-axis position, rather than always rotating to A0, allowing probing of stock center at arbitrary A-axis angles. (PP-3128)
  - In PathPilot HUB, some older PCNC 1100 mill configurations were unable to switch to RapidTurn mode because of an INI file error. (PP-3133)
  - The alarm level of G-code errors detected at load was too low, making these errors less visible to users. This issue was introduced in v2.7.0/v2.7.1. (PP-3153)
- We slightly reduced the acceleration for microARC, and we reduced the G0 rapid speed from 20 rpm to 18 rpm. (PP-3144)

Mills

- We fixed issues where:
  - On Series 3 and older 1100 mills, program load/redraw reported a cryptic error message (program execution was unaffected). (PP-3120)
  - Mill conversational DXF failed to draw a preview for a positive Z End position. (PP-3125)

Lathes

- We fixed issues where:
  - For some metric thread pitches in conversational lathe tapping, the exported value didn't exactly match the desired value (for example, 1.4925 mm instead of 1.5 mm). (PP-3147)
  - The high belt spindle speed on 8L lathes was off by factor of 2. (PP-3151)

RELEASE NOTES FOR PATHPILOT V2.7.1

November 2020

Fixed Issues

Mills

- We resolved an issue, introduced in v2.7.0, which had the potential for certain error conditions using abort syntax to be ignored. Specifically, probing and some ATC errors were affected. (PP-3115)

Lathes

- We resolved an issue, introduced in v2.7.0, which had the potential for certain error conditions using abort syntax to be ignored. Specifically, automatic collet closer 'not clamped' detection before spindle start may not have functioned correctly. (PP-3115)

Routers

- We resolved an issue, introduced in v2.7.0, which had the potential for certain error conditions using abort syntax to be ignored. Specifically, the spindle chiller alarm wouldn't be reacted to properly. (PP-3115)

RELEASE NOTES FOR PATHPILOT V2.7.0

November 2020

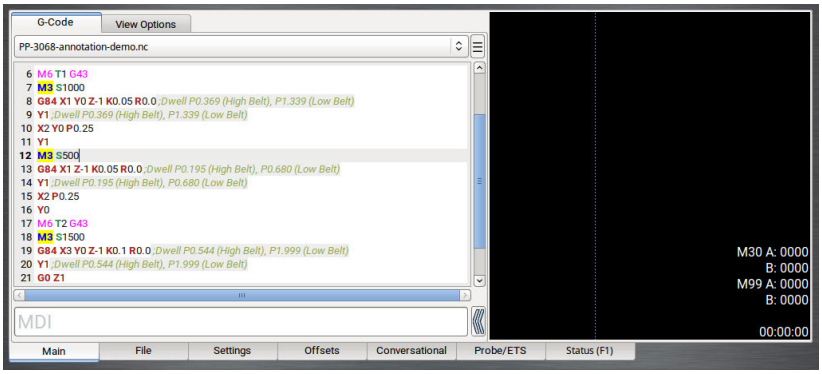
Enhancements

All

- For more consistent behavior with other computers, you can now abort a button click after pressing the mouse by moving off of the button before releasing the mouse. (PP-1852)
- G30 motions in a part program are now shown in the toolpath preview. (PP-2845)
- We added support for date and time stamping in G-code-based log outputs with a new #<_epochtime> parameter (which converts the current UTC epoch time in seconds). (PP-3006)
- We improved error detection in `G84` / `G33.1` rigid tapping that were previously reported as warnings. (PP-3029)
- We improved error checking in rigid tapping to allow shallower tap cycles. (PP-3043)
- G-code errors from program load now highlight the line causing the error in the G-code source view, and scroll to the error location. (PP-3045)
- We updated the caution message displayed on the **ETS Setup** tab to provide more detail on how to set the G37 position. (PP-3053)

Mills

- For G84 non-rigid tapping, an annotation is now shown in the source view with the calculated dwell time for the given spindle speed/belt position. This makes fine-tuning the dwell time easier. (PP-3049)



- We improved G-code colors in the Source View and editor, including previously missing G-codes, some comment syntax, M3/M30 ambiguity, and others. (PP-3013)
- The ATC set tool change position now logs old and new positions. (PP-3055)
- When setting the tool change position, the ATC now displays the acceptable range in metric if G21 is enabled. (PP-3056)
- We improved the usability of A-axis center probing with an option to find stock center without rotating the A-axis, and a visually distinct probe model in the live plot. (PP-3060)
- We added 0.5 seconds dwell between ATC drawbar activation and spindle raise. (PP-3093)

Fixed Issues

All

- We fixed issues where:
  - PathPilot opened a file with no extension in the editor rather than loading it as a program. (PP-3023)
  - Using the feature to find the A-axis center and set the origin (on the **Probing Rect / Circ** tab) didn't store the correct offset values. (PP-3059)
  - **Find A-axis center** (on the **Probing Rect / Circ** tab) sometimes returned incorrect values when used in G21 mode.
  - In some cases, the tool path live plot wasn't cleared if the tool view was filtered. (PP-3063)
  - If the controller was rebooted without coming out of reset, the current tool in spindle would not be restored the next time the **Reset** button was selected. (PP-3097)
  - Several buttons in the mill ATC and lathe tool touchoff tabs didn't respond after pressing the **Enter** key, needing an explicit mouse click instead. (PP-3098)

Mills

- We added support for A-axis positioning moves (part of traverse move to initial position) with G84 (PP-3014), and A-axis support for G30. (PP-3061)
- We fixed issues where:
  - G84 soft tapping cycles had an extra delay at the bottom of the hole, which could compress the tension-compression head on retract. (PP-3018)
  - Axis couldn't be referenced without the enclosure door closed if door switch/lock was installed. This behavior should have been limited to only CE configurations. (PP-3019)
  - In rare cases, G84 soft tapping generated a cryptic error message in long-running programs. (PP-3028)
  - A status message that displayed when the tool change position was out of range didn't accurately display the range. (PP-3057)
  - The A-axis display checkbox in the **View Options** menu could be unchecked while a program was running. The display would be unchanged, but the checkbox no longer matched the display state. (PP-3065)
  - Conversational circular pockets very close to the tool diameter would do a straight plunge instead of a helical ramp. (PP-3050)
  - Conversational rectangular pockets could be cut oversize in X if the tool radius was very close to the corner radius. (PP-3052)
  - Conversational rectangular/circular pockets could cause a plunge entry to extend past the -X edge of the pocket. (PP-3067)
- We updated the Fusion 360™ post-processor (provided by David Loomes), which has the following changes:
  - Added support for partial circular bore and partial circular boss probe operations.
  - Implemented size and position tolerance checking for the stock (set the **wrong size** or **Out of position** check boxes on the **Actions** tab for each probe operation in Fusion).
  - Option to print probe results to the status screen (set the **Print Results** check box on the **Actions** tab).
  - Implemented the **Measure tool** and **Tool break control** options in **Manual NC** CAM operations.
  - Implemented **Tool break control** when selected on the **Post Processor** tab in the Fusion 360 tool library.
  - Provided multiple options to control retraction at various stages during program execution. This replaces the old **Use G30** and **Use G28** options to give finer control. Default settings give the same results as Use G30 = Yes and Use G28 = No.

Tip! For more information on the changes made to the Fusion post-processor, we recommend watching David Loomes' YouTube video: https://youtu.be/qrJZmE5qgzI

Lathe

- Previously, conversational programs posted `G30 Z#5422`. This meant that G30 didn't move in X, even if the **G30 Move in Z Only** checkbox was cleared (on the **Settings** tab). Now, conversational programs post only `G30`. (PP-2930)
- We fixed an issue where the **Spindle RPM** slider in CSS mode wasn't uniform, reaching maximum spindle speed before the 100% position. (PP-3016)
- We added error detection to prevent G33.1 rigid tapping in G96 mode and protect against tap breakage. (PP-3042)
- We fixed an issue where set start line on the lathe didn't correctly track tool offsets. (PP-3087)

Plasma

- PathPilot now immediately recognizes a failed plasma arc and pauses machine motion until the torch can be relit. (PP-2999).

RELEASE NOTES FOR PATHPILOT V2.6.0

August 2020

Enhancements

All

- We added support for the 1300PL machine. (PP-1377)
- We added PathPilot HUB file transfer capabilities to the **File** tab. (PP-2613)

Mills

- We extended and improved the A-axis display in the **Tool Path** display, which makes it easier to understand the effective tool path relative to the workpiece. (PP-2862)
- On the **Settings** tab, there's now a **4th Axis Rotary** sub-tab. We consolidated the 4th axis rotary settings and configuration on this new tab. (PP-2894)
- G64 Naive CAM Detection now supports rotary axes, which smooths out 4th axis Fusion 360 programs. (PP-2907)

Fixed Issues

All

- We fixed an issue where there was a missing unit conversion in incremental polar coordinates. Previously, incremental polar angles were interpreted as radians instead of degrees. (PP-2905)
- We fixed a rare case with G95, where the first feed move immediately following a G33 move would use the wrong feed rate. This didn't occur if G33 was followed by other motions like G0, tool change, etc. (PP-2923)
- When you typed G or A in the **MDI Line** DRO field (for valid G-code commands), the auto-complete dropdown displayed. Now, you must type two characters for the auto-complete dropdown to display. (PP-2928)
- We fixed an issue with conversational edit where an edited coolant setting would be ignored if an operation was edited again before saving. (PP-2965)
- We fixed an issue with non-Tormach (unsupported) touch screens, where a calibration error occurred when the controller powered on. (PP-2980)
- We fixed an issue where the background grid for the front/side G-code view in the **Tool Path** display was incorrectly displaced when large tool offsets were active. (PP-2986)
- The time zone for Virtual PathPilot controllers on HUB is now set according to the user's HUB account preferences. (PP-3004)
- When you add a .txt file to PathPilot, you can now either load the file as G-code or view and edit it with the text editor. Previously, .txt files **only** opened in the text editor. (PP-3005)

Mills

- We fixed an issue, introduced in PathPilot v2.3.6, that prevented 4th axis homing from working. (PP-2955)
- We fixed an error in a Fusion 360 probing routine (f360_probing-y-channel.ngc) that incorrectly used x instead of y. (PP-2956)
- We fixed an error where, when you tried to set a reference height with a tool other than Tool 0 on the **Probe Setup** tab, the green LED came on after selecting **Set Reference Height** (which falsely indicated that the position was set). (PP-2991)
- We fixed an issue where, in the tool table, either missing tools (between tool 1 and tool 256) or duplicate tools (between tool 257 and tool 1000) weren't correctly displayed. (PP-2997)

Lathe

- We fixed an issue where, after re-editing an existing file that used conversational threading, the operation didn't update to reflect the number in the **Passes** DRO field. (PP-2925)
- We fixed an issue that prevented conversational lathe threading from using the maximum allowed number of threading passes (98). (PP-2964)

RELEASE NOTES FOR PATHPILOT V2.5.2

June 2020

Fixed Issues

All

- Typing `ADMIN` in the **MDI Line** DRO field displays a list of possible commands. We fixed an issue where using a mouse to click one of those commands could also click an item behind the list (like buttons in the **Persistent Controls** section, or lines of code in the **G-Code** window). (PP-2904)

Mills

- We fixed issues where:
  - The 4th axis appeared to stall when it ran a G93 line, because it incorrectly interpreted the F word as the number of minutes (rather than 1/[number of minutes]). (PP-2914)
  - The feed rate was too large with G93 and some G1 moves (with large A-axis motion and very small X, Y, or Z motion). (PP-2917)
  - Improvements in PathPilot v2.5.1 to G43 unintentionally made some actions on the **ATC** tab unreliable.

Lathe

- In some situations, the set start line feature failed to run with programs that used G33/G76 threading cycles. This has been fixed. (PP-2920)
- We removed the tool path's green **Refresh** button, because the tool path now automatically refreshes. (PP-2921)
- You can now use the **100% Override** buttons (for feed rate, spindle rpm, and maximum velocity) during G-code execution. (PP-2922)

RELEASE NOTES FOR PATHPILOT V2.5.1

May 2020

Enhancements

- We optimized the tools used to draw the tool path preview to reduce the loading time for G-code programs. (PP-2583)
- You can now keep subroutines in the same folder as the .nc file. Previously, they had to be in a **subroutines** folder. (PP-2697)
- On M+ and MX mills and the 24R router, you no longer have to set the Electronic Tool Setter (ETS) nose reference after each time you reference the Z-axis. (PP-2742)
- We rearranged the contents of the **ETS Setup** tab (on the **Probe** tab) to improve the user experience. (PP-2768)
- A G30 command now moves the Z-axis before moving the X- and Y-axes . Previously, it did a single, coordinated move of the X-, Y-, and Z-axes. (PP-2778)
- To measure tool lengths using an Electronic Tool Setter (ETS), we added a **Z only** checkbox to the **ATC** tab. (PP-2807)
- Selecting a line in the **G-Code** tab now highlights the corresponding section of the part's geometry in the **Tool Path** display (and vice versa). (PP-2833)
- When you open a .txt file, it now opens in the text editor. Previously, it tried to load as a G-code program. (PP-2835)
- We installed the Realtek 8812AU driver, which is required for certain WiFi adapters. (PP-2839)
- The Electronic Tool Setter (ETS) Z minus speed is now limited so that, in the case of an unexpected ETS trip, it travels a maximum of 4 mm before coming to a controlled stop. (PP-2864)
- When you reload a G-code file, it now retains your previously set G-code view options. Loading a new G-code file still resets the default options. (PP-2854)
- We reworked the **Rect/Circ** tab (on the **Probe** tab):
  - We changed the buttons from **Find Center and Set Origin** to **Find Center**, and we added radio buttons to control when to set the origin and when to just display the results. This means that you now can find the center without also changing the origin. (PP-2856)
  - We added a DRO field that displays the dimensions of the most recently used probe function, like the diameter of a circular boss or the distance between two X positions. (PP-2580)

Mills

- We added support for CE-marked M and MX mills.
- To adjust the operator console's touch screen sensitivity in humid environments, we added the command **Admin Touchscreen Sensitivity**. (PP-2855)

Router

- You can no longer use an M4 (spindle reverse) command in a G-code program, which could cause premature wear on certain machine components. (PP-2838)

Fixed Issues

- We fixed an issue where probes retracted from the effective probe coordinate (with tip radius compensation) rather than the actual probe tip coordinate. (PP-2792)
- If the probe input changed during a probing move, PathPilot incorrectly displayed a message attributing the error to a tool setter input change. We edited this message. (PP-2800)
- We fixed an issue where Fusion 360 probe subroutines didn't correctly apply extended work offsets in G21. (PP-2885)
- When using a CNC scanner, the **Post to File** button on the **Scope** tab (within the **Scanner** tab) didn't work. This has been fixed. (PP-2867)

Mills

- Previously, when the spindle stopped on a PCNC 440, there was a long delay before it restarted. We shortened the pause. (PP-2777)
- On the **Settings** tab, in the **Accessory Input** section, we edited the text on the **Active Probe** selection to include the Electronic Tool Setter (ETS). (PP-2804)
- Previously, conversational pocketing routines had a divide by zero exception. This has been fixed. (PP-2878)
- We fixed an issue where the 4th axis selection (on the **Settings** tab) persisted through power cycles, but the 4th axis scale settings didn't correctly apply when the machine powered on. (PP-2861)
- On machines with a microARC 4 installed, adjusting the **Maxvel Override** slider changed the speed values to those used with the 6 in. and 8 in. 4th axes. The values for the correct 4th axes are now retained. (PP-2891)

RELEASE NOTES FOR PATHPILOT V2.4.3

March 2020

Enhancements

All

- We reorganized the options for set start line on the **G-Code** tab. The most commonly used option is now listed first. (PP-2820)

Fixed Issues

All

- We fixed issues where:
  - In the **G-Code** tab, files with an M30 command included at the end of the program didn't automatically scroll back to the first line. (PP-2830)
  - Selecting either **Stop** or **Reset** didn't affect the playback of videos in an M01 comment. (PP-2819)
  - In some situations, selecting **Set start line (no preparation)** from the **G-Code** tab could have applied incorrect tool offsets. (PP-2813)
  - Creating a G-code file from a .dxf file added tool changes after M03 and M08 commands rather than before. (PP-2811)

Mills

- We fixed issues where:
  - G84 tapping cycles occasionally stalled after the spindle reversed directions. (PP-2818)
  - When using the **Spindle Override** slider, the **RPM** DRO field sometimes displayed a value higher than the machine's maximum rpm. (PP-2808)

RELEASE NOTES FOR PATHPILOT V2.4.2

February 2020

Enhancements

All

- To increase visibility, we added a blue background to messages in the Tool Path display (like when a tool change requires operator help, or M00/M01 comments). (PP-2806)
- We improved the stability of USB devices (like a keyboard or a mouse). (PP-2776)
- Starting the program in the middle with start line now accurately restores machine state as though it had executed the program up to that line. Additionally, you now have three options for control over lead-in moves. For information, see "Set a New Start Line" (page 43). (PP-2691)
- We added several MDI commands to help you more easily navigate large G-code files. (PP-2731) New MDI commands include:
  - `GOTO [line-number]` and `N [line-number]`: Scrolls to the specified line in the G-code.
  - `SET START [line-number]`: Scrolls to the specified line in the G-code and sets it as the start line with no lead-in move.
  - `SET START [line-number] LINEAR`: Scrolls to the specified line in the G-code and sets it as the start line with a linear lead-in move.
  - `SET START [line-number] PLUNGE`: Scrolls to the specified line in the G-code and sets it as the start line with a Z plunge lead-in move.
- You can now access a calculator when you type the command `ADMIN CALC` into the MDI Line DRO field. (PP-2764)

Mills

- We improved Conversational pocketing:
  - Rectangular pockets are now specified by their center point using the pattern table on the Drill/Tap tab. This means that you can now define multiple rectangular pockets in one Conversational step (like you can with circular pockets).
  - We adjusted spiraling feeds, which were previously slow, to use the feed DRO.
  - Tighter geometries are now supported. If a pocket is too small for a spiral entry strategy, PathPilot now uses a zigzag or plunge entry.
  - We enhanced chip clearing by raising the tool between cutting sections of the geometry.
- To avoid rare situations in which using non-Tormach probes resulted in an error ("Probe is already tripped when starting G38.2 or G38.3 move"), we increased the probe retract distance. Previously, for a typical probe tip diameter of about 0.118 in., the retract increased from about 0.020 in. to 0.059 in. Now, the retract distance is half of the probe tip diameter. (PP-2793)

- We added support for the BT30 spindle upgrade kit, which adds all MX functionality to an M+ mill. (PP-2773)
- If X or Y tool offsets are applied, the Tool DRO field now displays in italics. You can hover over the field to read the offset value. (PP-2726)
- The Set Reference Height button on the Probe Setup tab now includes an LED light. When it's lit, it indicates that the probe length reference height has been set. (PP-2690)
- We improved Z depth control while rigid tapping on MX mills (peck tapping). (PP-2608)
  For example:

```
M3 S500
G84 X1 Y2 Z-.3 R.1 K0.1
Z-.6
Z-.9
Z-1.2
(etc)
```

- You can now use an optional `I` word with G33.1 to increase the retraction speed while rigid tapping on MX mills. (PP-2552)
- For threading and rigid tapping on MX mills, the thread start position is consistent over a wider spindle speed range. (PP-2549)
- We added an `ADMIN ENCODER TEST` command, which verifies the installation and operation of the spindle encoder on MX mills. (PP-2424)

Lathe

- The default rapids are now 100 ipm. To decrease the maximum velocity, we added an `ADMIN RAPIDS` command. (PP-2728)

Fixed Issues

All

- We fixed issues where, in some situations, selecting Stop from the PathPilot interface caused the following issues:
  - Incorrectly altered the G-code modal machine state. (PP-2762)
  - Didn't highlight the current line in orange. (PP-2761)
- We fixed an issue where, if a G-code file had G37 programmed on the same line as M6, the tool offset didn't correctly apply. (PP-2746)
- For touch screens that have the X and Y sensor axis swapped, we improved the calibration process. (PP-2752)
- Previously, using conversational editing removed any unavailable tabs from the PathPilot interface. Now, the tabs are shown as disabled. (PP-2578)

- We fixed issues where:
  - Aborting a program during execution incorrectly applied G92 offsets. (PP-1262)
  - Using conversational engraving to engrave a serial number of 0 failed. (PP-2751)
  - In rare situations, stopping program in middle of tool change sometimes displayed a dialog box that required the user to add or remove tools from spindle. (PP-2730)

Mills

- We fixed the following Conversational DXF issues:
  - After loading a .dxf file, the generated G-code file ignored any changes to DRO values on the left side of the Conversational tab (like Tool or Spindle RPM). (PP-2796)
  - Coolant and the spindle are no longer turned off and back on in between every shape in the generated G-code file. (PP-2795)
  - DXF to G-code now correctly supports 0 rpm. This means that machine motion won't stall while it's waiting for the spindle-at-speed signal. (PP-2794)
- Previously, if you re-edited a .dxf to G-code conversational file, but the .dxf file had been moved, PathPilot displayed an error message. Now, a dialog box displays from which you can navigate to the new .dxf file location. (PP-2781)
- We fixed issues where:
  - When using Fusion 360 probe subroutines, the probe's tool path preview didn't match the actual run-time tool path. (PP-2747)
  - The probe length setup procedure in G21 didn't work correctly. (PP-2765)
  - When switching probe or tool setter types, if the verification dialog was canceled, later probe or tool setter trips caused PathPilot to exit unexpectedly. (PP-2755)
  - While using a high-speed spindle on a PCNC 1100 mill, typing any value into the RPM DRO field resulted in an error. (PP-2753)
- We improved the decision making that's made when the Automatic Tool Changer (ATC) moves to the tray in position. (PP-2719)

RELEASE NOTES FOR PATHPILOT V2.4.1

December 2019

Fixed Issues

Mills

- We fixed issues where:
  - On the **ATC** tab, the **Touch Off Entire Tray** button didn't run the command as expected. (PP-2732)
  - Using `G0/G1` to retract a probe resulted in a motion error. This did not affect probing routines. (PP-2736)
  - While in `G21` mode, setting a `G37` position didn't work. (PP-2738)
  - If an M+ or MX machine was already powered on and out of reset, bringing it out of reset a second time (after pressing and releasing the Emergency Stop button) resulted in the following error: **EMC_AXIS_INCR_JOG**. (PP-2739)
  - In a few rare cases, tool motion stopped during a 3D profiling pass. (PP-2740)
  - While in `G21` mode, setting ETS Work Offset (height) and ETS G37 Spindle Nose Reference didn't work correctly. (PP-2741)

RELEASE NOTES FOR PATHPILOT V2.4.0

December 2019

Enhancements

All

- On the Main tab, the Tool Path display now shows M30 and M99 cycle counters. (PP-1703)
  For information, see "Use Cycle Counters (M30 and M99)" (page 128).
- We added support for G37 and G37.1, used to automatically measure tool lengths with an Electronic Tool Setter (ETS). (PP-2682)
  For information, see the most recent version of the machine operator's manual at tormach.com/support.
- We added support for the 24R router. (PP-2702)
- You can now use the command `ADMIN AUDIO` in the MDI Line DRO field to adjust your audio device and volume. (PP-2706)
- In the MDI Line DRO field, commands now autocomplete if you first type the word `ADMIN` or `FIND`. (PP-2707)
- You can now manually refresh the Tool Path display by selecting Refresh (in the Position Status area). Previously, it refreshed automatically, which could sometimes cause interruptions to your workflow. (PP-2709)

Mills

- Feed rate override functions are now disabled when a probe is completing its final (slow/fine) motion in the subroutine. (PP-2705)

Lathe

- When using conversational threading, you can now add a value for spring passes. (PP-1299)

Fixed Issues

All

- On the Settings tab, the G-Code Description window now shows entries for G61, G64, and extended work offsets. (PP-2663)

- We fixed an issue where conversational feeds and speeds were enabled when the PathPilot controller powered on, even if the checkbox on the Settings tab was cleared. (PP-2664)

Mills

- We fixed issues where:
  - The Accessory Input LEDs between the Status tab and the Probe tab didn't consistently show the port's actual state. (PP-2425)
  - A bounce on probe input between G38.2 and G38.6 incorrectly stopped the program and caused an error: "Probe tripped during non-probe motion." (PP-2495)
  - RapidTurn configurations had an incorrect maximum rpm for the high spindle speed range. Now, the maximum is 3500 rpm. (PP-2628)
  - The PathPilot controller wouldn't restart if the engraving_fonts folder was missing (for example, if you deleted it). (PP-2654)
  - The conversational Engrave tab wasn't compatible with extended work offsets (G54.1 P1 or larger). (PP-2661)
  - On 770MX and 770M+ mills, RapidTurn configurations had incorrect scales for Y- and Z-axes. (PP-2696)
  - In probing an A-axis circular boss, the final Z probe for center was above the actual center by the probe effective tip radius. (PP-2718)
  - When using the pendant to jog in G21 mode on a PathPilot operator console, the machine moved in inches (instead of millimeters). For example, instead of moving 0.1 mm per click, the machine moved 0.01 in. per click. (PP-2721)
- We modified the limits available to use with the soft limit commands (like `ADMIN SET_X_LIMIT`). (PP-2585)
- We updated the Fusion 360™ POST (provided by David Loomes), which was modified to work with extended work offsets. (PP-2695)

Lathe

- We fixed an issue where, when using a roughing tool for a conversational finishing pass, the spindle could stall. (PP-2651)

RELEASE NOTES FOR PATHPILOT V2.3.6

October 2019

Fixed Issues

All

- We fixed issues where:
  - Stopping a program early sometimes applied inconsistent tool or work offsets, which persisted until a new offset was applied. (PP-2669)
  - Typing M99 in the **MDI Line** DRO field without any G-code files loaded caused PathPilot to fail until the controller was restarted. (PP-2671)

Mills

- We fixed issues where:
  - Changing the probe rate DROs in G21 (metric units) resulted in a double conversion and unexpected results. (PP-2662)
  - Selecting **Find Z-** (on the **Probe** tab) failed if the start point for the Z-axis was below the machine's Z- soft limit. (PP-2666)
- The physical buttons for feed hold and cycle start on the PathPilot Operator Console now respond when they're pressed, not when they're released. (PP-2670)

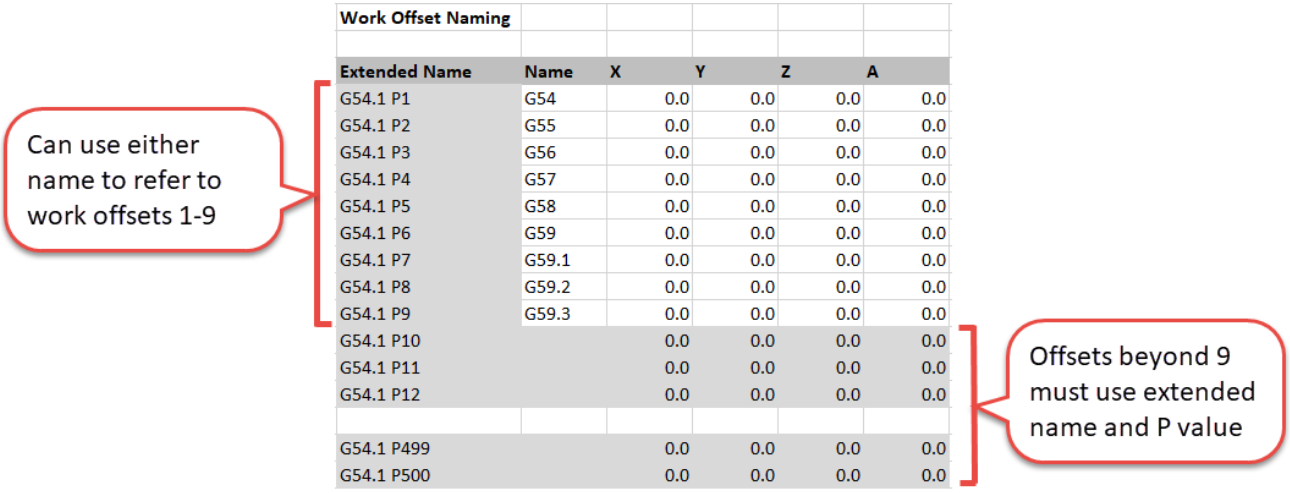
RELEASE NOTES FOR PATHPILOT V2.3.4

October 2019

Enhancements

All

- We increased the number of work coordinate systems from 9 to 500. (PP-2439)
  You can access the extended work offsets by using G54.1 with a P word value of 1 to 500. G54.1 P1 through P9 refer to the same work offsets as the classic G54, G55,..., G59.3 codes, as shown in the following figure.



- You can now add videos to display in an M00 or M01 break by adding the file name (which is case sensitive) as a comment after the M-code. (PP-2500)
- From the Main tab, you can now quickly edit a G-code program with conversational programming by either (PP-2539):
  - Using a new keyboard shortcut: Shift+Alt+E.
  - Selecting **Show in File tab** by right-clicking inside the G-Code tab or selecting the Options menu.
- We reduced the sensitivity of the Zoom slider on the View Options tab for touch screen users. (PP-2562)
- We added support for 1100MX and 770MX mills. (PP-2592)
- We added support for xsTECHrouter bench-top router. (PP-2593)
- You can now create a new G-code file from PathPilot: from the Settings tab, make sure that no file is selected, and then select **Edit G-Code**. (PP-2610)

Mills

- You can now use M07 for mist coolant on base M machines with the second J5 relay. (PP-2499)
- You can now toggle the spindle speed range between high and low during a manual tool change operation. (PP-2589)

- In the Probe Setup tab, we added an Effective Tip Diameter section. This means that:
  - Probe feed rates are no longer controlled by the main feed rate DRO field. Instead, use the feed rate DRO fields on the Probe Setup tab as shown in the following figure.

  - You can use PathPilot to measure the effective probe tip diameter:
    1. Position the probe tip inside a known diameter, like a ring gauge or bearing race.
    2. In the Ring Gauge Dia DRO field, type the known diameter.
    3. Select Move/Set Tip Diameter.
       The probe measures the inside diameter of the ring and computes the effective probe tip diameter.
    4. In the Effective Tip Dia DRO field, read the value displayed. This value is also set in the tool table entry for the probe (tool 99).

  - We added support for G74 (left-hand threading) and G84 (right-hand threading) tapping cycles. Depending on the machine, G74/G84 adapts its behavior to the provide the best tapping available. For example, on an MX mill, it'll use the encoder for feedback with rigid tapping; on all other machines, it'll depend on timing with a tension/compression tapping head.
    The following words are supported with G74 and G84:

Word	Meaning
X	Tapped hole X position

Word	Meaning
Y	Tapped hole Y position
Z	Nominal Z depth
R	Z retract height
P	Dwell time (ignored for rigid tapping)
K	Thread pitch in units per thread
F	Feed rate (indirectly specifies thread pitch along with spindle speed)

To more easily use the K word to specify imperial threads, you can:

- Do simple math in the G-code. For example, to program a 10 TPI thread:

  ```
  G84 ... K[U_INCH[1 / 10]]
  ```

  The U_INCH operator treats the value in brackets as a value in inches and, if G21 is active, it's automatically converted internally as necessary.

- Use a new U_TPI operator. For example:

  ```
  G84 ... K[U_TPI[10]]
  ```

Peck tapping is also supported. For example:

```
G0 Z1 (start at a safe clearance height)
G99 G84 X1 Y2 Z-.1 R.1 K0.125
Z-.2
Z-.3
...
Z-1.0
G80
G0 Z1 (back to clearance height)
```

- On MX mills with a RapidTurn or High-Speed Spindle installed, PathPilot now automatically switches VFD modes. This means that MX mill users don't need an SD card to switch between milling operations and RapidTurn or High-Speed Spindle operations.

Lathe

- We updated the referencing behavior on lathes and RapidTurn so that it's consistent with mill configurations. You can now select the buttons one right after another, and once the machine references one axis, it'll move on to the next. (PP-2591)

Fixed Issues

All

- We updated the jogging behavior (especially with a jog shuttle wheel) and user interface so that it's consistent across mill, lathe, and RapidTurn configurations. (PP-1427)
- We fixed issues where:
  - When creating conversational jobs in metric mode (G21), the job assignment editor indicated that jobs were unsaved even if they were unchanged. (PP-2487)
  - If the probe input bounced between G38.2 and G38.6, an error occurred: "Probe tripped during non-probe motion." (PP-2495)
  - In some situations, attempting to jog the machine with the jog shuttle while PathPilot was loading a file sometimes resulted in difficult-to-stop axis motion. (PP-2566)
  - In some situations, PathPilot became unresponsive after selecting the Exit button, and then selecting OK in the dialog box with an E-stop reminder. (PP-2575)
  - Running over 500 G-code files or performing more than 200 conversational edits without powering off PathPilot could result in failures, sometimes with a "Too many open files" error. (PP-2640)

Mills

- We fixed issues where:
  - When switching between the Probe and Status tabs, the accessory input status LEDs displayed inconsistent states. (PP-2425)
  - When creating a rectangular face in conversational programming, the spiral count was sometimes off by one. (PP-2493)
  - On M, M+, and MX mills, pushing in the Emergency Stop button didn't turn off the coolant if it was already on. (PP-2641)

Lathe

- We fixed an issue where, when loading a G-code file, an error could occur regarding M3 or M4 aborting due to collet clamping status. (PP-2586)

RELEASE NOTES FOR PATHPILOT V2.2.4

July 2019

Enhancements

All

- The G-Code tab now only highlights (in orange) the line of code that PathPilot is currently executing. In single block mode, after completing one line of code, it highlights the next that will execute when you select Cycle Start. (PP-2515)
- We simplified the use of tool path history lines in the Tool Path display. Now, when you select Cycle Start, the lines remain if you set a start line other than the first line of the G-code file. In all other cases, the lines are cleared when you select Cycle Start. (PP-2519)
- The tool path history now reflects the tool selections made on the View Options tab in the G-Code window. (PP-2523)
- We improved reliability when updating G-code files from another computer over the network in the following situations:
  - When you save a new version of the currently loaded G-code file, and it's replaced on the controller. (PP-2524)
  - When you save a new version of the currently executing G-code file, and it must be reloaded on the controller. (PP-2525)
- We improved the keyboard shortcuts for the MDI Line DRO field (PP-2540):
  - If a command has an error and the Status tab opens, you can now use Alt+Enter to return to the MDI Line DRO field.
  - If you're done typing commands, you can now select the Enter key to remove the focus from the MDI Line DRO field (for example, to jog the machine using the keyboard).

Fixed Issues

All

- We fixed issues where:
  - While in RapidTurn mode, the Update button was unresponsive. (PP-2486)
  - When changing to tool 0 and selecting M6 G43, the tool length offset of the previously used tool remained. (PP-2488)
  - Error messages related to the Tool Table window didn't display in the correct units. (PP-2491)
  - A program that ended in M30 didn't scroll the G-Code tab back to the first line of code after the program stopped. (PP-2513)

- When a newly applied tool offset changed the Z position, yellow (jogging) history lines appeared in the Tool Path display, even though no motion had occurred. (PP-2520)
- We improved calibration on touch screens that we haven't tested or added support for. (PP-2516)

Mills

- We fixed issues where:
  - If the hard stop referencing was disabled (because of unusual fixture requirements) when using a RapidTurn on an M+ mill, unexpected machine table movement could occur while the mill was referencing the Z-axis. (PP-2484)
  - The appropriate 4th axis option didn't persist after it was selected. (PP-2517)
  - When using coordinate rotation, the Tool Path display didn't immediately redraw with the changed rotation. (PP-2537)
  - While using a 4th axis with an M+ mill, the 4th axis couldn't be jogged before referencing the X, Y, and Z axes. (PP-2541)

Lathe

- We fixed issues where:
  - In some situations, starting a G-code program at a line greater than 1 could result in unexpected motion from feed rate motion commands earlier in the file. (PP-2528)
  - During G33 spindle-synchronized motion (threading), a warning wasn't displayed to indicate if the required feed rates exceeded the machine's maximum feed rate. (PP-2545)

RELEASE NOTES FOR PATHPILOT V2.2.2

May 2019

Fixed Issues

All

- We fixed an issue where using the Admin Logdata command didn't copy the log files .zip to the USB drive if the drive name had a space in it. (PP-2479)
- We improved the soft travel limits for each axis: when you enter a value, the soft travel limit range is now validated. You can also now view the current values for any soft travel limits on the Status tab. (PP-2483)

Mills

- We fixed issues where:
  - The following machines (with shared X- and Y-axis limit switches) sometimes incorrectly attempted to reference both axes at the same time (PP-2477):
    - PCNC 1100 and PCNC 770 mills with an Enclosure Door Switch Kit installed
    - PCNC 440 mills
  - In some situations, reverting to a previous version of Automatic Tool Changer (ATC) firmware failed. (PP-2481)
  - Probe input wasn't triggering on PCNC 440 mills. (PP-2482)

RELEASE NOTES FOR PATHPILOT V2.2.0

April 2019

Enhancements

All

- After you press the Stop button, the G-Code window now displays the last line that PathPilot executed. (PP-1755)
- You can now update PathPilot or transfer log files (for troubleshooting) to a USB drive from the Tormach Machine Controller Configuration screen. (PP-2358)
- You can now try to calibrate any touch screen, even if we haven't tested it or added support for it. (PP-2364)
- The Tormach Machine Controller Configuration screen now displays images and descriptions of each machine to help you make the correct selection. (PP-2427)

Mills

- You can now override the feed rate or spindle speed by up to 200%. Previously, the maximum was 150%. (PP-2275)
- In addition to Z offsets, G43 now applies X and Y offsets. (PP-2402)
- We added support for 1100M+ and 770M+ mills. (PP-2420)
- We added support to the ATC firmware for a BT30 spindle. (PP-2429)
- We included G-code subroutines for Fusion 360™ probing (provided by David Loomes). (PP-2440)
  For information, go to the PathPilot folder **f360_probe**, and open the file **README.txt**.
- We added a Show Tools drop-down menu above the Tool Table window. (PP-2464)
  You can now filter tools based on:
  - Tools used in the currently loaded G-code file
  - Tools with descriptions
- We added a Search field above the Tool Table window to improve search functionality for tool descriptions. You can now type searches in any letter case, search for any word in the description, or search for other tool characteristics (like tool number, diameter, or length). (PP-2465)

Lathe

- PathPilot now confirms that the collet is clamped when it reaches an M3 or M4 command. Previously, it confirmed when you selected Cycle Start. This means that you no longer need to clamp the collet before you select Cycle Start. (PP-2399)

Fixed Issues

All

- We fixed issues where:
  - By default, Samba shares with Windows 10 didn't work. (PP-1858)
  - If a folder had a space in its name, the file chooser displayed that there was no free space in that directory. (PP-2342)
  - Selecting Set Start Line in a G-code program was off by one line. For example, if you set the new start line to line 6, the program started on line 5. (PP-2346)
  - Using the Admin Settings Backup command didn't include tools 256 through 1000 from the Tool Table window. (PP-2397)
  - PathPilot displayed the G-code status after the controller powered on, but before you initially selected Reset. (PP-2457)
- When the enclosure doors are open, we changed the default maximum spindle speed to 0 rpm. (PP-2433)
  To change the maximum spindle speed:
  - In the **MDI Line** DRO field, type `ADMIN OPENDOORMAXRPM [MAXIMUM SPINDLE SPEED]`.
- If you're working with a G-code program that's larger than 100,000 lines, you can now load all tool path preview lines. (PP-2455)
  To load all tool path preview lines:
  - From the **Main** tab, on the **G-Code** tab, do one of the following:
    - Right-click anywhere in the window. Then, select **Load All Preview Lines**.
    - Tap anywhere in the window. From the **Options** menu, select **Load All Preview Lines**.

Mills

- We fixed issues where:
  - An M7 command didn't actuate a MultiCool mist on 1100M or 770M mills. (PP-2355)
  - In some situations, SmartCool stopped responding to G-code commands. (PP-2393)
  - Canceling a tool change on a mill with no Automatic Tool Changer (ATC) installed displayed error messages for an ATC. (PP-2396)
  - The RapidTurn configuration for an 1100M mill used 9.5 in. of Y-axis travel rather than 11 in. of Y-axis travel. (PP-2405)
  - When creating a routine in Conversational using the same tool, the Z-axis didn't move to the clearance location between pocket routines. (PP-2415)

Lathe

- We fixed an issue where, after a door open error, the spindle didn't start through a command in the MDI Line DRO field. (PP-694)

RELEASE NOTES FOR PATHPILOT V2.1.6

December 2018

Enhancements

All

- You can now read information about all available G-code modes by hovering over the items in the G-Code Description window on the Settings tab. (PP-2258)
- If there's an available network adapter, PathPilot now finds it as the controller powers on. To improve the out-of-box experience, we added an alert offering to use it to set up the network. (PP-2292)
- We maximized the file storage on the PathPilot controller's internal drive by growing the partition and file system in-place. This results in a program storage increase of over 75 GB on most controllers. (PP-2301)
- Warnings for zero-length tools (on a mill) or tools with the default tip orientation 9 (on a lathe) are now suppressed if a file of the same name is loaded. (PP-2304)
- In Conversational threading, the Quick Reference drop-down now lists an option for SAE 5/16-18. (PP-2309)
- When an axes' work offset value is changed, the Status tab now displays a message with the old and new values. (PP-2314)
- We changed the label above the axes' Work Offset DRO fields to display the current work coordinate system as a reminder. (PP-2315)

Mills

- We added support for the **Door Lock Switch Kit (PN 38283)**. (PP-2166)
- When coordinate rotation is active, the text in the X-axis Work Offset DRO field and Y-axis Work Offset DRO field now displays in an italic font. (PP-2307)
- After enabling an **Enclosure Door Switch Kit (PN 35550)**, an LED displays on the Status tab. For clarity, we changed its label from Door Switch to Door Open. (PP-2311)
- If air pressure is momentarily below the minimum requirement while using an Automatic Tool Changer (ATC), tool change reliability is now improved. (PP-2326)

Fixed Issues

All

- We fixed issues where:
  - A G-code file with blank lines before the first % line sometimes didn't display the image or comment in a M00 or M01 break. (PP-2259)
  - G90.1 and G91.1 didn't display in the Status read-only DRO field. (PP-2298)
  - The View Options tab didn't expand correctly when the G-Code window was expanded. (PP-2302)

- We added support for Dropbox for Business accounts. The sync folder now appears on the File tab using the Dropbox for Business team's name. (PP-2303)

Mills

- We fixed issues where:
  - PathPilot ignored door switch functionality while in RapidTurn mode. (PP-1757)
  - Manually changing a tool with an Enclosure Door Switch Kit required you to select Cycle Start twice in order to resume program execution. (PP-2312)
  - Loading a large .dxf file sometimes timed out with a dbus error. (PP-2313)

RELEASE NOTES FOR PATHPILOT V2.1.5

October 2018

Enhancements

All

- On the Main tab, we added a View Options tab. You can now:
  - View a selective Tool Path display by tool numbers. (PP-2014)
  - Do things like change the view of the Tool Path display (iso, top, front, or side), change the resolution of the grid lines, or zoom in and out on the Tool Path display while using a touchscreen. (PP-2285)
- To indicate lack of motion or unusual levels, the Feed Rate Override, Spindle Override, and Maxvel Override sliders now turn yellow when they're either at 0% or above 100%. (PP-2281)
- Each file chooser now displays the amount of free space on the drive. (PP-2295)
- On the Status tab, we added a LOG DATA button. During support requests, you can now use the button to quickly collect diagnostic data. (PP-2297)

Mill

- You can now load G-code programs or switch between G20 and G21 modes faster. (PP-2291)

Fixed Issues

All

- We improved support for more Tormach touchscreens. (PP-2200)
- If you're using a G-code file that PathPilot has run before (and saved an estimated time), the time remaining clock now displays immediately after the file loads. This means that you no longer need to select Cycle Start to view the estimated time remaining. (PP-2257)
- We fixed an issue where the Tool Table window was sometimes off by one row when auto-scrolling to match the current tool number. (PP-2265)

Mills

- We fixed an issue where earlier mills (Series 3 and before) couldn't reference a 4th axis with or without the limit switches enabled. (PP-2254)
- On 1100M and 770M mills, we adjusted the VFD feedback tolerances to prevent stalls while waiting for spindle at speed. (PP-2282)
- We improved Automatic Tool Changer (ATC) tray alignment reliability in some situations of continual tray rotation in one direction. (PP-2284)
- We fixed an issue where, if the spindle started manually in forward or reverse, the RPM DRO field didn't provide feedback that included the Spindle Override slider. (PP-2293)

Lathe

- In lathe conversational profiling:
  - We improved the feature to eliminate gouging in some situations. (PP-2270)
  - We fixed an issue where a G-code file could post that had leading zeros in sub reference error. (PP-2273)
  - We fixed an issue where, in some situations, retracting could result in a transit through the stock. (PP-2276)

RELEASE NOTES FOR PATHPILOT V2.1.4

September 2018

Enhancements

All

- On the File tab, added functionality to cut, copy, and paste files (PP-995)
- During program execution, added a time remaining clock (which displays above the time elapsed clock) based on the cycle's previous run time (PP-1500)
- The first time Cycle Start is selected for a program, PathPilot now provides a warning message if the program uses tool table entries that haven't been configured (PP-1673)
- Tools used by the current G-code program are now indicated in the Tool Table window — if a tool is used, its corresponding Tool column header becomes shaded (PP-2013)
- On the File tab, the USB Files window now automatically navigates down to the root folder of the USB drive (PP-2059)
- When not using a USB drive, the File tab now resizes to better use screen space (PP-2177)
- Improved logging (PP-2181):
  - After a program stops, the most recently executed G-code line number is now logged in case a program must be restarted from the middle
  - Cycle time and run time are also logged — the run time is the cycle time minus the time that the program paused at line M00 or M01
- Added two new admin commands to adjust the timing of tooltips (PP-2182):
  - `ADMIN TOOLTIP DELAYMS` sets the milliseconds prior to displaying the tooltip (and then again for the expanded tooltip). The default is 1200 milliseconds.
  - `ADMIN TOOLTIP MAXDISPLAYSEC` limits the amount of time the expanded tooltip displays. The default is 15 seconds.
- Added an `ADMIN MOUSE` command to adjust mouse preferences, like pointer speed and right- or left-hand button mapping (PP-2186)
- Added an options menu to the G-Code window so that, while using a touchscreen, you can do things like set the start line, or control the colors of line numbers and G-code (PP-2211)
- To comply with CE standards, we changed the colors of some buttons and LED lights (PP-2224)
- In the logfiles folder, there's now a log file with a history of G-code programs that have been run and their timings, called **gcode_log.txt** (PP-2237)

- Improved support for `LOGOPEN` and `LOGAPPEND` G-code commands, so that the resulting log files are now visible on the File tab (PP-2239)

  Note: For information, see the LinuxCNC documentation.

- If the current program file is changed and reloaded, the override sliders now maintain their set value (PP-2250)
- The Status tab is now visible while a G-code program is running, so that you can view diagnostic information (PP-2251)

Mills

- Added Y tool offsets to the lathe interface when using a RapidTurn (PP-1519)
- Improved the response time when selecting the Conversational tab if there are many font files in the engraving_fonts folder (PP-2170)
- On the Engrave tab, added search capability to the Font list (PP-2172)
- Changed the electronic tool setter (ETS) finish touch off feed rate to 2.5 in./min, regardless of the initial feed rate (PP-2201)

Fixed Issues

All

- We fixed issues where:
  - A very brief limit switch trigger put the machine into reset, and didn't display an error message on the Status tab (PP-1671)
  - When using M98 for looping, the current and next highlighted lines in the G-Code window were sometimes incorrect (PP-1975)
  - The Font list didn't correctly scroll with keyboard navigation (PP-2171)
  - If a font was removed or added to PathPilot while the conversational programming's Engrave tab was visible, the currently selected font wasn't preserved (PP-2174)
  - After renaming a file or folder name, the newly named file or folder wasn't automatically selected (PP-2180)
  - If the onscreen keyboard was displayed when a file was changed, the prompt to reload the file was hidden, so it looked like the interface was locked (PP-2188)
  - When posting a file using conversational, the title wasn't used as the default file name (PP-2207)
  - Changing from G20 to G21, followed by a G30 command, could have resulted in an incorrect location used for G30 (PP-2208)
  - When frequently using the Conversational Edit dialog, PathPilot performance slowed (PP-2215)

- G7x canned cycles didn't obey exact path mode during cycles (PP-2238)
- When setting the work offset (by selecting a Zero button, or entering a command in a DRO field), the entry didn't immediately update the Work Offsets table (PP-2244)
- If the work offset was changed, the display in the Tool Path window didn't immediately redraw (PP-2245)

Mills

- We fixed issues where:
  - A circular pocket less than two times the diameter of the tool couldn't be created using conversational programming (PP-1987)
  - When importing a .csv tool table, the diameter was divided by two (PP-2205)
  - When using an automatic tool changer (ATC), a tool pick up position could be set before referencing the Z-axis (PP-2221)
  - If tool parameters were changed directly by a G10 command, they didn't display correctly in the Tool Table window (PP-2240)
  - If changes were made to tools 1 through 256 in the tool table, and the controller unexpectedly restarted, the changed values weren't retained (PP-2233)
  - After selecting Start Scan, the scanner sometimes took minutes to start (PP-2219)
  - In 770M RapidTurn configurations, there were rare amplifier fault errors (PP-2223)
  - Tool number 1000 couldn't be entered into the Tool DRO field, because it was limited to three digits (PP-2209)

Lathe

- We fixed issues where:
  - The manual tool change message didn't clear from the Tool Path window after selecting Cycle Start (PP-2191)
  - In some conditions, the lathe G7x remap code generated exceptions and errors on load (PP-2229)
  - The command `ADMIN RESET_SOFT_LIMITS` didn't correctly function (PP-2126)
- We now consistently treat pressing **ESC** on the keyboard as a shortcut for the Stop button in lathe and RapidTurn configurations (PP-1495)

RELEASE NOTES FOR PATHPILOT V2.1.3

July 2018

Fixed Issues

Lathe/RapidTurn

- We fixed an issue that was introduced in PathPilot v2.1.1 where constant surface speed (CSS) wasn't working correctly in both G96 and the **CSS** DRO field (PP-2199)

RELEASE NOTES FOR PATHPILOT V2.1.2

July 2018

Fixed Issues

Mills

- When using an ATC, we fixed issues where:
  - The interface was unresponsive throughout an operation to touch off the tool tray, making it difficult to stop the operation (PP-2056)
  - Certain programmed feed rates could time out an operation to touch off the tool tray (PP-2194)
- On M Series mills, we fixed an issue where the variable frequency drive (VFD) speed was incorrectly reported if the VFD wasn't powered on before an M3 or M4 command (PP-2189)
- On PCNC 440 mills, we fixed an issue where the VFD status was sometimes inaccurately detected, which incorrectly prevented Automatic Tool Changer (ATC) operations (PP-2193)

RELEASE NOTES FOR PATHPILOT V2.1.1

June 2018

Enhancements

All

- Added an option to import and export the tool table as a .csv file (PP-1906)
- When editing conversational job assignments, the G-code preview now uses the same syntax highlighting and line numbering as the G-code preview on the **Main** tab (PP-2091)
- If a touch screen is installed, the calibration utility is automatically started during the initial setup (PP-2120)
- Added tooltips to provide detailed information for many areas of the interface (PP-2130)
- To make it easier to update the software, added a network name to the file name of the automatic settings backup (PP-2135)
- If the network name is changed, the network file sharing is now automatically restarted, and doesn't require a power cycle (PP-2136)
- Improved the diagnostic ability when using more than one USB M-Code I/O Interface Kit (PP-2142)
- To see the time, you can now click the PathPilot logo. Right-click the time to adjust it if it's incorrect. (PP-2169)

Mills

- We expanded the tool table to allow up to 1000 tools (PP-1913)
- In the **Probe** tab, there's now an option to probe the inside corner of a part (PP-2011)
- We added support for 1100M and 770M mills (PP-2134)

Fixed Issues

All

- We added error messages that display if a USB M-Code I/O Interface Kit is enabled, but not properly connected or detected, to prevent starting a G-code program with integration M-code that's dependent on the USB M-Code I/O Interface Kit (PP-839, PP-1530)

- We fixed issues where:
  - By default, Samba shares with Windows 10 didn't work (PP-1858)
  - M98 sometimes called the subroutine fewer or more times than the L word called for (PP-1969)
  - Our error suppression logic hid a warning on the toolpath preview (PP-2073)
  - The **Zero Height Gauge** button was missing from the **Offsets** tab, so we restored the button for use with a height gauge (PP-2093)
  - In rare situations, trajectory planning didn't properly fall back to parabolic blending, which resulted in a brief pause (PP-2104)
  - When using a touch screen, it briefly lost calibration during a software update (PP-2119)
  - When editing conversational program files, G59.n work offsets were reset to G59 (PP-2187)
- We now consistently treat the **ENTER** key on an extended keyboard (next to the numeric keypad) the same as the normal **ENTER** key (PP-2102)
- In the **Tool Table**, we added search functionality for tool descriptions (PP-2144)
  To search for a tool description:
  1. On the **Tool Table**, click a tool number. Then, type the desired text to search.
     A search box appears, and the **Tool Table** scrolls to the first tool description that matches the search.
  2. *(Optional)* Use the **UP ARROW** or the **DOWN ARROW** to move between search matches.
  3. Press **ENTER** on the keyboard to stop searching.

Mills

- When using an Automatic Tool Changer (ATC), we fixed issues where:
  - After the first tool change, the **Find Rectangular Boss Center** routine from the **Probe** tab failed (PP-1777)
  - After manually changing a tool, the tool assignment wasn't removed from the tool tray, even if indicated (PP-2050)
- We fixed issues where:
  - When using conversational programming to make a pocket routine, the feed rate was sometimes output as zero (PP-1998)
  - When using an A word, a 360 degree G2 move had a feedrate lower than the F word (PP-2085)
  - When using DXF in conversational programming, a Z move was sometimes commanded before the first tool change and a G43 offset was applied (PP-2121)
- We added support for an alternate USB interface chip that's used with an ATC (PP-2179)

Lathe

- In conversational profiling, we fixed an issue where an error was received if the profile wasn't complete to the edge of the stock (PP-2103)

RELEASE NOTES FOR PATHPILOT V2.0.4

April 2018

Fixed Issues

Mill

- Fixed additional issues in relation to the new PCNC 440® Automatic Tool Changer (ATC) USB interface chip.

RELEASE NOTES FOR PATHPILOT V2.0.3

April 2018

Fixed Issues

All

- In the **Tool Table** window, we fixed an issue where pressing **ENTER** on the keyboard's numeric keypad added a new line instead of saving your edits (PP-2010)

Mill

- The USB interface chip for the PCNC 440® Automatic Tool Changer (ATC) was discontinued, so we added support for its replacement (PP-2081)

RELEASE NOTES FOR PATHPILOT V2.0.2

March 2018

New Features

For more information on added features, refer to the machine's operator manual at tormach.com/support.

All

- Added suggestions for feeds and speeds in conversational routines (PP-1090)

Mill

- In the **Conversational** tab, there's a **DXF** tab to import a .dxf file and generate G-code from it (PP-1930)

Enhancements

All

- To make it easier to understand the order of error messages, there's a time stamp added to the beginning of each message on the **Status** tab (PP-1794)
- When loading a .jpg, .jpeg, or .png file, the image viewer opens to make sure that you're loading the right file (PP-1823)
- When adjusting the override or jog speed sliders, you can use the scroll wheel on the mouse to make adjustments in 2-percent increments (PP-1900)
- If a conversational DRO field is highlighted red because of an error in the value entered, we added a context-sensitive tool tip to help you troubleshoot it (PP-1935)
- To reduce confusion, we moved the light that indicates the currently selected axis on the Jog Shuttle to the right of the **Axis** DRO field (PP-1959)
- To make it easier to read error messages, the **Error Message** window in the **Status** tab is automatically maximized based on your machine's configuration (PP-1960)
- If the text in a comment after M00 or M01 doesn't match an image file name, the comment now displays over the toolpath preview (PP-1983)
- If the **Feedrate Override** slider or the **Spindle Override** slider is set to over 100 percent, it's now reset to 100 percent when loading a G-code file (PP-2020)
- The percentage set on the **Jog Speed** slider is remembered through power cycles (PP-2021)

Mill

- In the **Probe** tab, there's a button to change the corner on which to probe a part (PP-1742)
- To improve usability, we expanded the size of the font selection list in the **Engrave** tab by removing the **Font Preview** window (now, the font preview applies to the text entered in the **Text** field); also, we fixed a bug in the Bebas font name (PP-1832)

- When an Automatic Tool Changer (ATC) is doing **Go to Tray Load** and **Remove** operations, ATC reliability is improved; also, we improved the interface's responsiveness (PP-1940)
- In the **ATC** tab, when deleting all tool assignments, there's an alert message to confirm the deletion (PP-2018)

Lathe

- In conversational, there's a tag in the G-code comments to indicate from which **X** side the tool is cutting (PP-1596)
- In conversational profiling:
  - In the lathe profiling points table, there's an alert message that displays after you click **CLR** to confirm the deletion (PP-1833)
  - There's an option to use only **Roughing** or only **Finishing** (PP-1865)
- In the **Settings** tab, there's support for the **Automatic Collet Closer Kit for 15L Slant-PRO (PN 38690)** (PP-1918)
- In the **Tool Table** window, editing table cells auto-advances to the next column (PP-2044)

Fixed Issues

All

- We clarified this error message: "Circular move on line 40 would exceed joint 0's negative limit" (PP-1767)
- We fixed an issue where, when editing G-code, the **Save As** dialog box was missing a field for the file name — which made it difficult to save a new version of a G-code file, or to create a new G-code file to save on the PathPilot controller (PP-1809)
- In the **Status** tab, when a limit switch is pushed in, the axes limit switch lights are now yellow to indicate an unusual status (previously, they were green); also, we added a context-sensitive tool tip to indicate if the limit switch is disabled (PP-1847)
- When in conversational editing, we fixed an issue where using **Save As** in some path and file name situations produced an unexpected exception (PP-1878)
- In conversational job assignment editing, we fixed an issue where **Insert Step** inserted below the current **File** tree row only (PP-1880)
- We fixed an issue where the spindle tip drawing was outside of the work envelope box by the tool offset value (PP-1882)
- We fixed an issue where using the **Spindle Speed Range** button during a programmed M00 or M01 break was prevented (PP-1883)
- When using the override and jog speed sliders, we fixed an issue where percentages were changed when **HOME** or **END** was pressed on the keyboard (PP-1901)

- We fixed an issue where changing the network name was only saved if **ENTER** was pressed on the keyboard (to view the new network name, you still must power cycle the PathPilot controller) (PP-1903)
- In the **Tool Table** window, we fixed an issue where using **LEFT ARROW** or **RIGHT ARROW** on the keyboard jogged the machine instead of moving the cursor between cells (PP-1939)
- If the custom metric or SAE threads files are deleted, the template for custom files is now automatically restored, which has helpful examples as comments (PP-2048)

Mill

- We clarified this ATC message that displays when removing a tool from the spindle: "Insert T0 in spindle" (PP-1851)
- In conversational external thread milling, we fixed an issue where entering values in DRO fields required a certain order (PP-1808)
- In conversational engraving, we fixed an issue where serial number engraving failed at a small height (PP-2015)

Lathe

- In conversational profiling, we fixed issues where:
  - An exception error was sometimes received when the profile was first loaded (PP-1724)
  - The **RPM** and **Feed** DRO fields were grayed out after selecting **Drill** or **Thread** tabs (PP-1796)
  - Arcs that continued from arcs with a different radii could cause errors (PP-1849)
  - Loading some G-code files caused errors (PP-1860)
  - The generated tool path could, in some circumstances, gouge the part (PP-1863)
  - A rapid occurred from **-X** to the far **+X** side of a part before cutting started (PP-1871)
  - A math overflow error, in some situations, caused a duplication of steps (PP-1872)
  - Editing a profiling step, and then posting the same profile, could produce slightly different G-code (PP-1873)
  - A finish pass could result in a part gouge for some series of lines and arcs (PP-1874)
  - A very small radius could fail before the finish pass (PP-1875)
  - Entering **0** in a DRO field in the radius column caused an error (PP-1931)

- We also fixed issues where:
  - After switching between conversational editing and loading G-code files, the tool path sometimes displayed in the wrong orientation (PP-1932)
  - PathPilot wouldn't exit if all three of the following conditions happened (PP-1951):
    1. A USB M-Code I/O Interface Kit was enabled
    2. The **Status** tab was active
    3. The **Emergency Stop** button was pushed

RELEASE NOTES FOR PATHPILOT V2.0.1

Release Date: January 9, 2018

Fixed Issues

All Machines

- Some files are not correctly recognized as G-code, and the **File** tab will not load them. (PP-1887)

Mills

- Resolved a RapidTurn® pausing issue when using G-codes dependent on spindle speed sensing.

Lathe

- Resolved a rare G96 arc blend following error.

RELEASE NOTES FOR PATHPILOT V2.0.0

Release Date: December 7, 2017

Enhancements

All Machines

- **M01** breaks are added after tool changes to G-code files programmed in Conversational (PP-1277)
- A **USB** button is added to the **Admin Settings Backup** and **Admin Settings Restore** dialog boxes (PP-1462)
- Before attempting a software update, checks for enough free disk space (PP-1498)
- Conversational includes M01 break at the end of generated code (PP-1521)
- Add a button to the **Internet** light on the **Status** tab for more obvious network configuration (PP-1551)
- G-code window on the **Main** tab uses the same syntax highlighting as gedit (PP-1613)
  Right-click in the G-code window to enable or disable.
- G-code window on the **Main** tab displays line numbers (PP-1614)
  Right-click in the G-code window to enable or disable.
- Normalized limit switch setting, and provided warning if they are disabled (PP-1662)
  The **Disable Home Switches** checkbox has been restructured — it is now an **Enable Limit Switches** checkbox.
- Add an **ADMIN MEMORY** MDI command (PP-1685)
- The override slider thumbnails are larger to make them easier to use with a touchscreen (PP-1701)
- Add tooltips to quickly see the full name for file and folder names that are shortened by an ellipsis on the **File** tab (PP-1702)
- Suppress warning dialog asking for an emergency stop prior to a software update if the machine status is already in an emergency stop (PP-1752)
- **Feedrate**, **Spindle RPM**, and **MAXVEL** override sliders feel sluggish while dragging thumb (PP-1758)
- Conversational Edit: M05, M09, and G30 commands are commented out between (mill) spot and drill operations (PP-1773)
- Apply patch for Wi-Fi WPA2 KRACK vulnerability (PP-1774)
- Implement a wait cursor during file copying operations for improved user feedback
- Add **ADMIN DROPBOX** MDI command and configuration tool

Mills

- Spindle **FWD** and **REV** buttons only function on the **Main** tab — like the **Cycle Start** functionality (PP-1624)
- On rectangular facing in Conversational, X and Y move first (PP-1695)

Lathe

- The **Tool** table size is increased from 26 entries to 99 entries (PP-1410)
- From the **Settings** tab, removed the **Keyboard Jogging Disable** check box (PP-1488)
- Make the lathe consistent with the mill in regards to G30 and the **Move in Z Only** checkbox on the **Settings** tab (PP-1705)
- Profiling is now supported in Conversational.

Fixed Issues

All Machines

- Shutdown screen: Instruction to power off computer is hard to read on some computers (PP-265)
- Configure gedit to show G-code syntax highlighting and display line numbers (PP-1219)
- Add detection and warning for attempting 2.x install on 1.x image (PP-1494)
- If a file name has spaces, the **ADMIN SETTINGS RESTORE** command does not work (PP-1520)
- Spindle **FWD** and **REV** should only function on the **Main** tab — like **Cycle Start** (PP-1554)
- exFAT formatted USB drives don't work well as targets for copy operations (PP-1583)
- Conversational Edit: Conversational generated file cannot be opened anymore by conversational code (PP-1594)
- G30 should check for reference before taking action (PP-1611)
- Reference offset increased from 0.010 to 0.025 to reduce chances of 'on limit error' when machine powers on (PP-1661)
- Logfiles: Double click **pathpilotlog.txt** tries to open it as a G-code program (PP-1672)
- Combo box of recently loaded G-code files offers files, even if they were deleted by the user (PP-1674)
- Warn on failures during copy or delete operations to **Status** tab, but keep going in multi-selection scenario (PP-1676)
- Conversational Edit: Adds multiple M01 commands when reconstituting a file with multiple steps (PP-1698)

Mills

- Probing: Probing buttons fail if axes soft limits are shortened through an **ADMIN SET_X_LIMIT** or **ADMIN SET_Y_LIMIT** command (PP-1199)
- Using 0 for depth of cut on thread milling conversational page and trying to post results in hung controller (PP-1683)
- 440 X+Y axis coasting after feedhold (PP-1694)
- Scanner: **Y Start** and **Y End** DRO field labels need to be swapped (PP-1763)


TITLE: externalThread-Mill
Aluminum
-any-
WORK OFFSET: G59
TOOL: 2
SPINDLE RPM: 555
FEEDRATE: 100.0
Z FEEDRATE: 100.0
Z CLEAR: 0.6000
(Flutes: N/A Chip load per tooth: N/A)
(SFM: 0 – MRR: N/A)
POST TO FILE
APPEND TO FILE
Face
Profile
Pocket
Drill / Tap
Thread Mill
Engrave
DXF (Mill)
EXTERNAL
INTERNAL
DEPTH
X (FROM DRILL TABLE)
Y (FROM DRILL TABLE)
NUMBER OF PASSES:
THREADS/IN:
3.0
Z START:
3.0000
Z CLEAR
PITCH (IN):
0.3333
2.0000
Z END:
LEFT
RIGHT
Main
File
Settings
Offsets
Conversational
Probe
Status
CYCLE START
SINGLE BLOCK
M01 BREAK
FEEDHOLD
STOP
COOLANT
RESET
100%
FEED 100%
100%
RPM 100%
100%
MAXVEL 100%
POS IN G55
DTG
ZERO X
5.1931
0.0000
REF X
ZERO Y
0.6512
0.0000
REF Y
ZERO Z
0.0000
0.0000
REF Z
ZERO A
0.0000
0.0000
REF A
STATUS:
G55 G90 G20 G80 G40 G94 G97 G99 G91.1
STEP:
REV
SPINDLE RANGE
T
0
05:06:02