

## Release Notes for PathPilot® v1.9.7

**Release Date:** September 20, 2016

**NOTE:** Updates do not need to be applied sequentially. Download the latest software update to be completely up-to-date on PathPilot operating system.

### Enhancements:

- #1265: Mill - Conversational: Improved tool path for circular, rectangular profile
- #1238: Changes to touch off for grooving and parting tool touch off. Previously, tool control point was set to the tailstock side when touching off; now, control point is set to the spindle side. All conversational routines have a tool width entry field to account for the change in control point. This was done in response to many customer requests to make touch off easier on grooving/parting tools.

**NOTE:** Old G-code conversationally created in versions before PathPilot v1.9.7 will not work with a tool touched off in v1.9.7 and later; new G-code conversationally created in v1.9.7 and all future versions will not work with tools touched off in versions v1.9.5 and earlier. If in doubt, touch off your parting/grooving tool again and repost the parting and grooving code.

### Bug Fixes:

- #1236: RapidTurn - Soft Limit: 770 RT configuration has incorrect soft limit travel setting for lathe Z-axis (mill X-axis)
- #1231: Mill - Conversational: Spot drill DOC DRO should accept either positive or negative values, then convert them to the value expected by the canned cycle
- #1238: Lathe/Rapid Turn: Selecting a parting tool on the Offsets tab defaults to tip orientation 1 or 4, but conversational parting routine requires 2 or 3. Workaround: manually edit orientation on Offsets tab.
- #1237: Mill - Conversational Thread Mill: Internal threads post with bad character on G64 line. Workaround: find G64 line and replace entire line with G64 P0.
- #1224: Mill - Conversational Thread Mill: Internal diameters with fine pitch (32 or greater) fail to interpolate helix because of G64 blending
- #1156: Mill - Touch Screen: Soft number pad doesn't work with drill table on conversational Drill/Tap tab
- #1223: Mill - Conversational Serial Number Engraving: Spindle RPM of 0 prevents G1 motion if engraving just a serial number
- #978: Mill - M8 or M9 on the same line as G40 gives error message
- #840: Mill - Interpreter: G50 and G80 should be allowed on same line



# Service Bulletin

---

## Release Notes for PathPilot® v1.9.6b

Release Date: July 12, 2016

### Bug Fixes:

- #1220: Mill - Conversational Engraving: Pressing Enter key after entering text in engraving text DRO results in “: Enter either Text and/or a Serial Number” error message
- #1210: Mill/Lathe - MDI: Don't remove focus from MDI line on non-ADMIN commands
- #1206: Mill/Lathe - \_sim\_specific.ini files missing in released .tgp file
- #1204: Mill - Error message when posting G-code with serial number on engraving tab
- #1205: RapidTurn - Conversational: Can't post drill G-code

**IMPORTANT!** This version of PathPilot removes Dropbox feature. For more information, refer to Tormach service bulletin SB0051: Uncommanded Spindle Acceleration in PCNC 440.

## Release Notes for PathPilot® v1.9.6

Release Date: June 27, 2016

### Enhancements:

- #965: Mill/Lathe - G-code window: Add 'Goto line number' and 'Search' utilities to right click menu
- #992: Mill - Conversational: Bolt hole circle generator on drill/tap tab
- #994: Mill/Lathe - Enhancement: Cycle counter and serialization macro
- #1198: Mill/Lathe - Add ADMIN RESET\_SOFT\_LIMITS command to MDI

### Bug Fixes:

- #1200: Mill/Lathe - Load G-code: G-code comment (1.006 10 fl) prevents loading of file
- #1189: Lathe - SFM DRO: If the spindle is running and a value of '0' is entered in the SFM DRO, the spindle speed will go to the maximum allowed speed
- #1187: Lathe - Conversational: Chamfer code does not use finishing feed rate, leaves roughing feed rate for finish pass
- #1158: Mill - Duality: "Commanded spindle speed outside of range" alarm is meaningless on Duality (manual spindle control), should not be shown to user
- #1197: Mill - Clear Current Program: CCP in G21 with a tool offset applied causes an odd, incorrect tool offset to be applied after the program is cleared
- #1185: Mill - ATC tab touchscreen keypad does not appear for INSERT or REMOVE DROs
- #1167: Mill/Lathe - Interpreter: Can't set start line in programs with numeric O program names (e.g., o1234)
- #1166: Lathe - Rigid Tapping: Rigid tapping is broken in v1.9.5.
- #1165: Lathe - Interpreter: G64 Blending has default tolerance that is much too large, results in rounding of sharp corners

**NOTE:** This software update may include a firmware update (depending on machine). If instructed to power cycle the controller after this update, please make sure to COMPLETELY remove power from the controller for at least five seconds before restarting the system.

## MDI FIND Command

MDI has the ability to search text of a G-code file for specific numbers, codes, or items of interest like tools, feeds, and speeds. Type *FIND* followed by the text to be searched in the MDI Line (see **Figure 1**). Pressing *Enter* finds the next instance of the searched text; pressing *Enter* while holding down the *Shift* key finds the previous instance. If found, PathPilot scrolls to the line containing the searched text and highlights it in yellow (see **Figure 1**). When the search reaches the end of the G-code file, it wraps and starts again from the beginning. Change the starting point of the search by clicking on any line in the G-code window.

## Special Searches

When used in conjunction with the *FIND* command, certain search terms (listed below) initiate a search through the G-code file to find more than just the actual search term:

- *FIND TOOL*: Searches for instances of the actual word *Tool* in the G-code and any *T* G-code command, which calls up a tool (e.g., T12)
- *FIND SPEED*: Searches for instances of the actual word *Speed* in the G-code and any *S* G-code command
- *FIND FEED*: Searches for instances of the actual word *Feed* in the G-code and any *F* G-code command (see **Figure 6.2**)

**NOTE:** Search text ignores case, so the command *FIND TOOL* will match *TOOL*, *Tool*, *tool*, etc.

The *FIND* command simplifies searching of a G-code file to verify speed and feed values and tool calls before cutting a part, or to find a specific set start line point in a large G-code file.

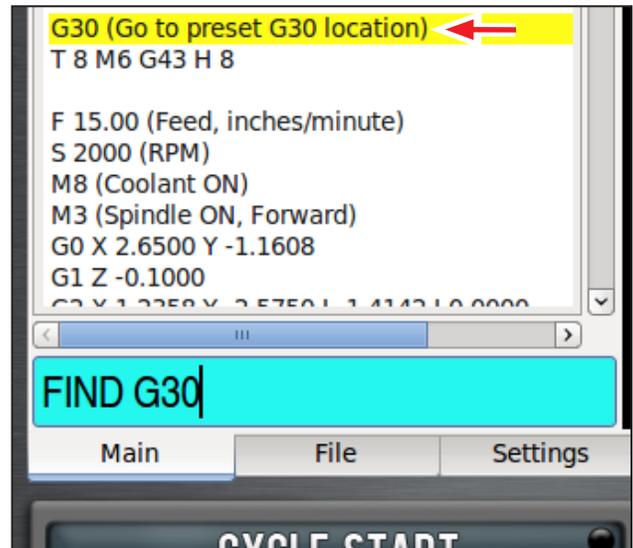


Figure 1

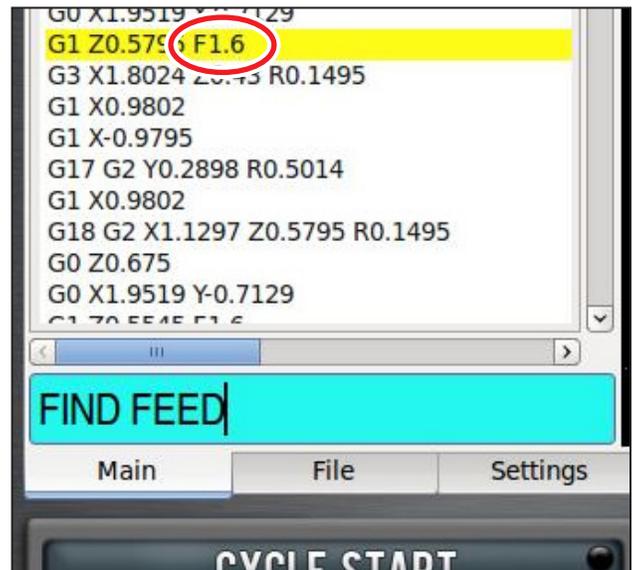


Figure 2

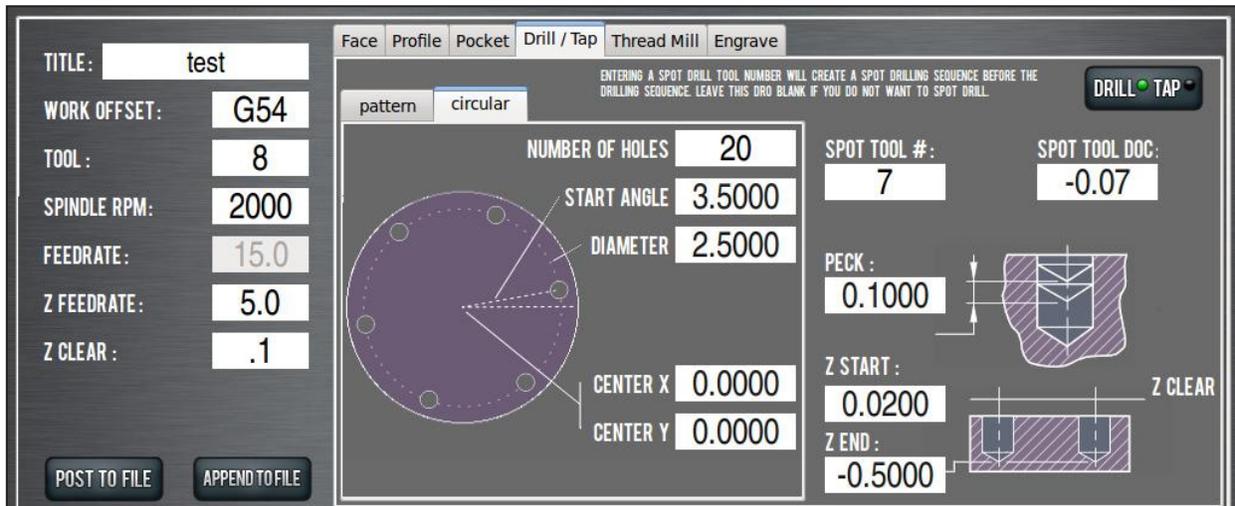


Figure 3

## Bolt Hole Circle Generator

The *Circular* tab creates a specific hole pattern of evenly spaced holes around a circumference, also known as a bolt pattern (see **Figure 3**). As with the *Pattern* tab, all the features and corresponding DROs, like *Spot* and *Peck*, are retained.

### Circular DROs

**Number of Holes** – Specifies the number of holes in the pattern. This must be greater than zero.

**Start Angle** – Specifies the angle from angle 0. Angle 0 is a base (horizontal) line from the center point going right (east) to the circumference. The angle from the base line can be either positive or negative, up to 90 degrees (or -90 degrees) and rotates the pattern either clockwise or counterclockwise. A negative angle produces a clockwise rotation; a positive angle produces a counterclockwise rotation. For example, to create a hex pattern with flats on the top and bottom, enter 0 into the *Start Angle* DRO. To create a hex pattern with flats on the left and right sides, enter 30 (or -30) into the *Start Angle* DRO.

**Diameter** – The size of the circular pattern as defined by a line through the center point of each hole.

**Center X, Center Y** – Defines the center point of the circular pattern

**Spot Tool #** – If this DRO contains a valid tool number when *Post To File* is clicked, a spot drilling sequence using this tool number will occur prior to the drilling sequence. The *Feedrate*, *Spindle RPM*, and *Z Clear* from the drilling sequence will be used for the spot drilling operation. The depth of cut for the spot drilling will be taken from the *Spot Drill DOC* DRO.

**Spot Drill DOC** – If the drilling operation includes spot drilling, this DRO will be used to determine the depth of cut for the spot drilling operation (refer to *Spot Tool #* section).

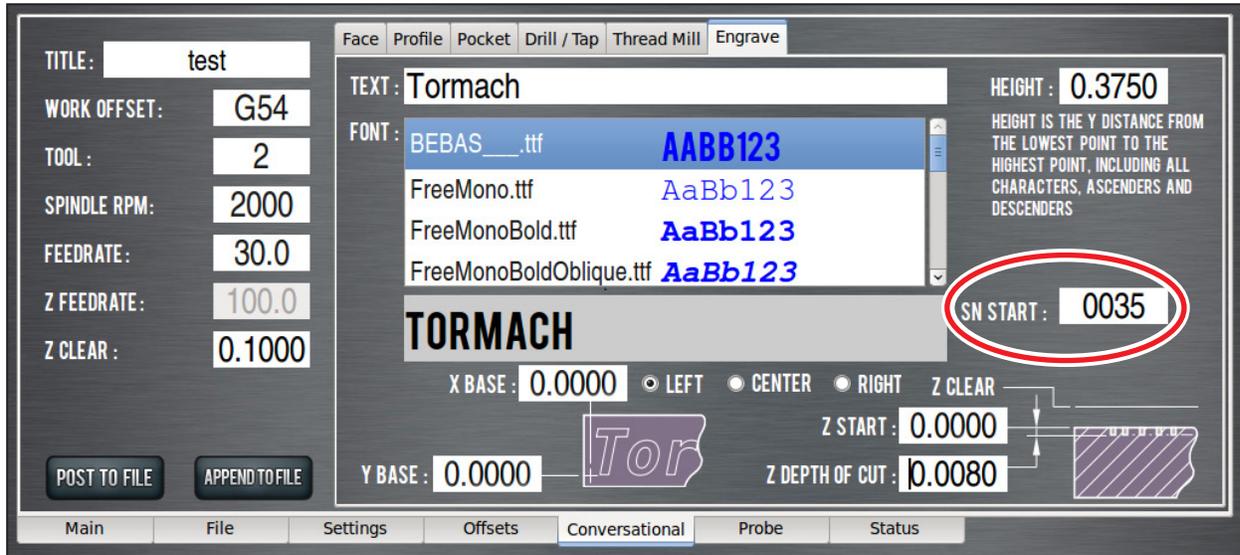


Figure 4

## Engrave Serial Numbers

Using the *Engrave* tab, serial numbers can be engraved alone or added to the end of any desired text – the number sequentially increases with each Cycle Start. Serial numbers use their own non-proportional font and are scaled to match the defined font extents.

**SN Start DRO** – Sets a starting serial number. Add zeros in front of the first digit as a hint to the number of decimals to be engraved in the series (including leading zeros). For example, if '0012,' '0013,' '0014,' '...' will be engraved. If '99' is entered, '99,' '100,' '101,' '...' will be engraved. The current serial number is stored internally; to view, hover over the *SN Start* DRO.

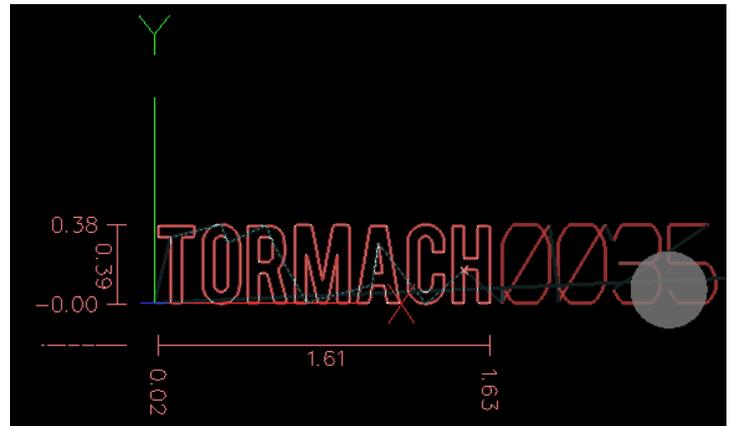


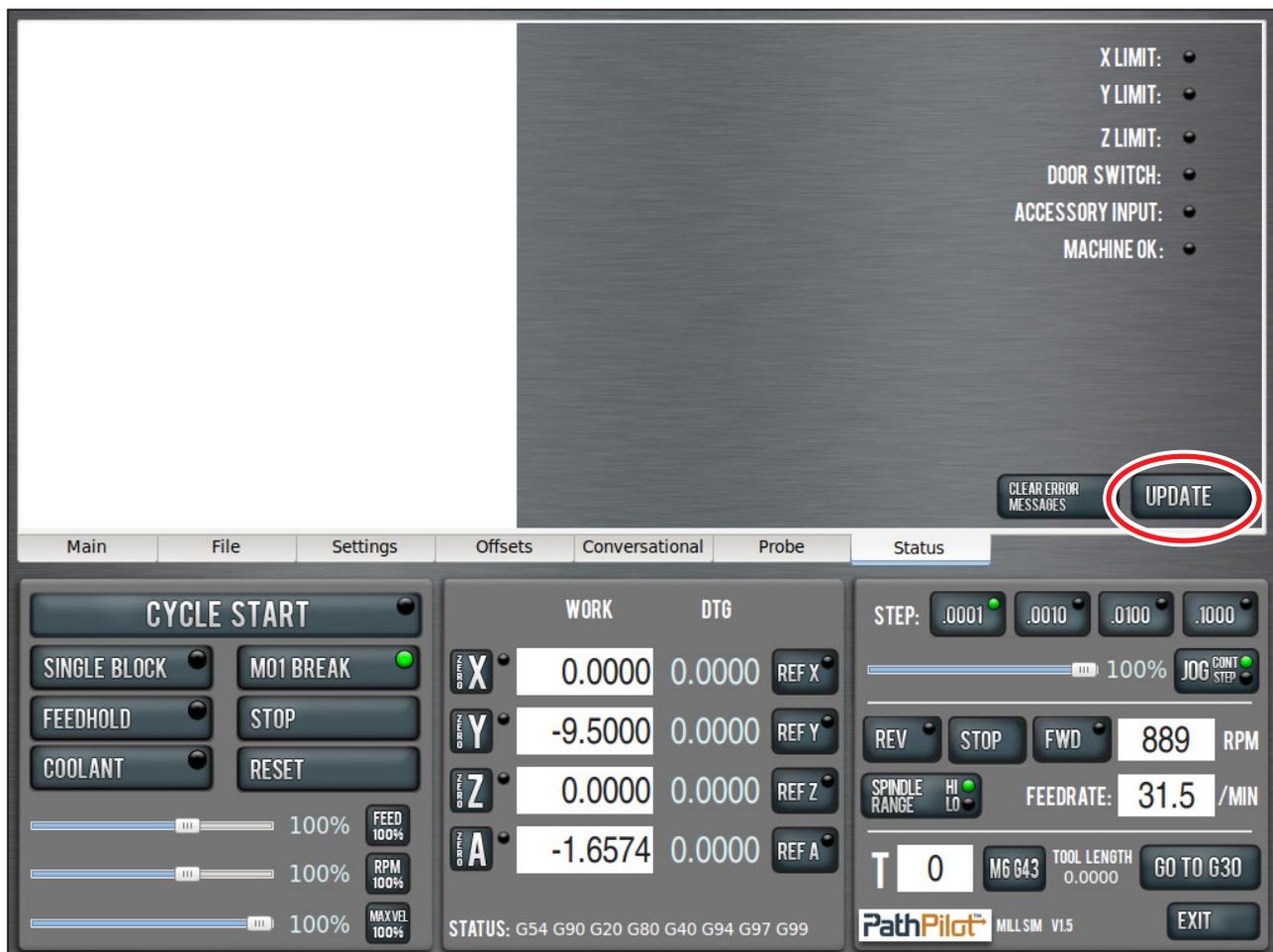
Figure 5

**NOTE:** Leave the Text DRO blank to only engrave a set of sequential serial numbers; likewise, leave SN Start blank to only engrave a line of text. It is an error if both the Text DRO and SN Start are blank.

## Software Update Instructions

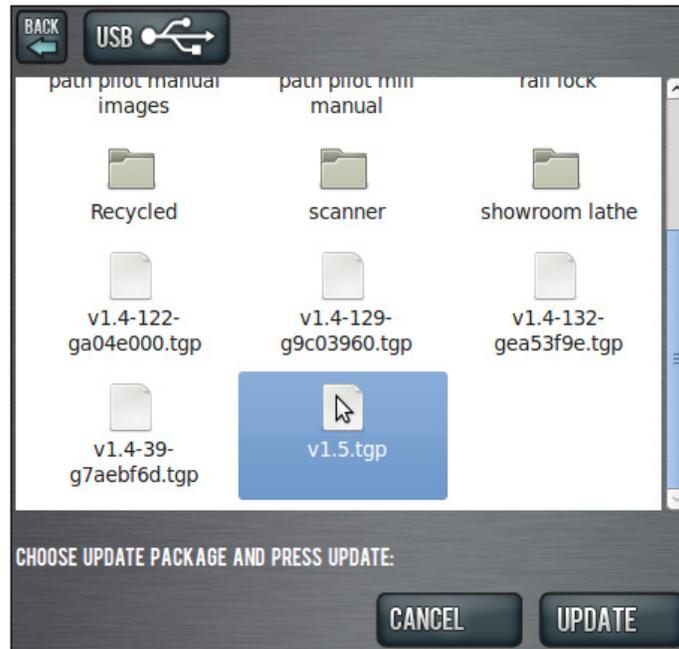
**NOTE:** Updates do not need to be applied sequentially. Simply download latest update file to be completely up-to-date on PathPilot operating system.

1. Download most recent PathPilot update file (vX.X) at [www.tormach.com/updates](http://www.tormach.com/updates)
2. Transfer file to a USB drive.
3. Put USB drive in PathPilot controller.
4. With PathPilot controller on and out of *Reset*, go to *Status* screen and click *Update* (see **Figure 6**).



**Figure 6**

5. Navigate to the (vX.X) file on USB drive; click *Update* (see **Figure 7**).



**Figure 7**

6. Push red E-stop button when prompted; click *OK* (see **Figure 8**).



**Figure 8**