



Service Bulletin

Release Notes for PathPilot® v1.9.6b

Release Date: July 12, 2016

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

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.

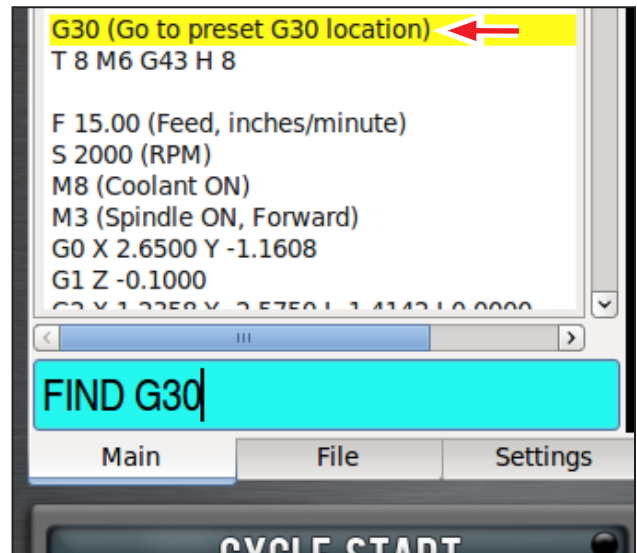


Figure 1

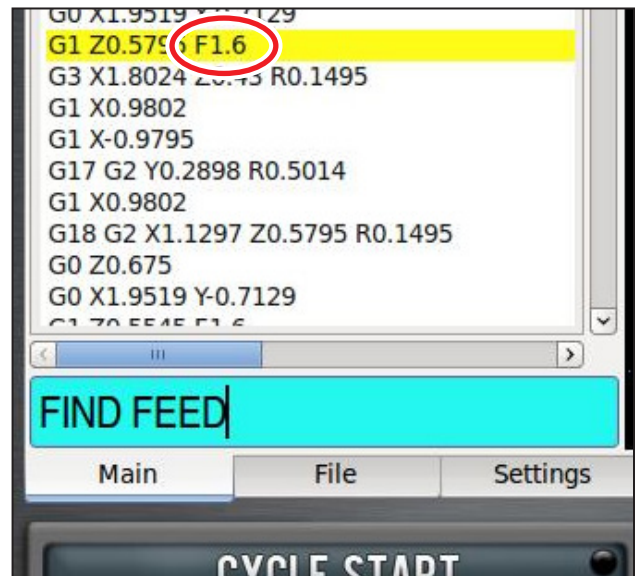


Figure 2

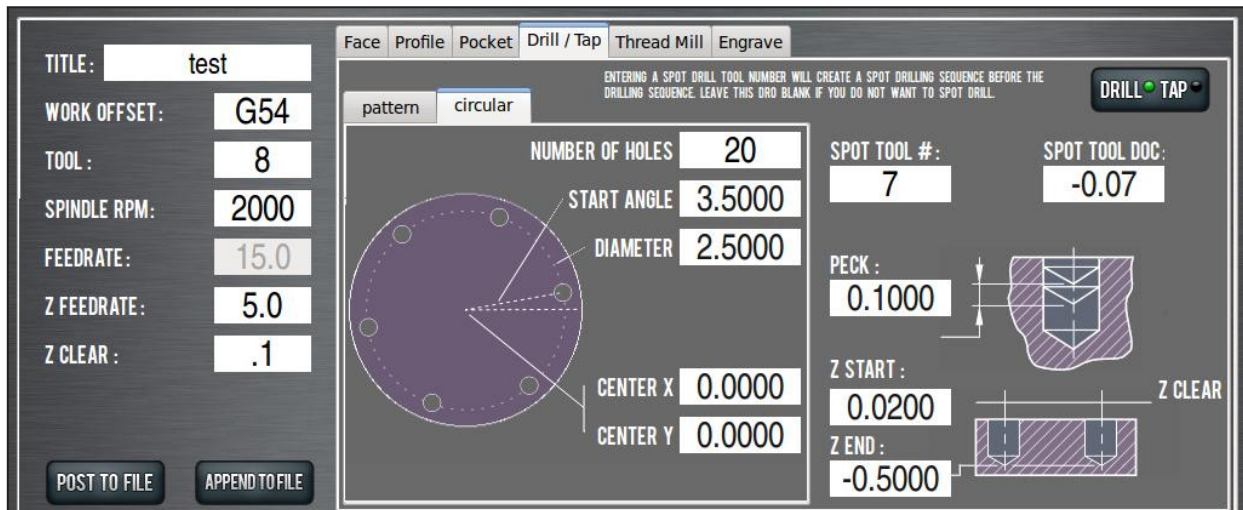


Figure 3

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).

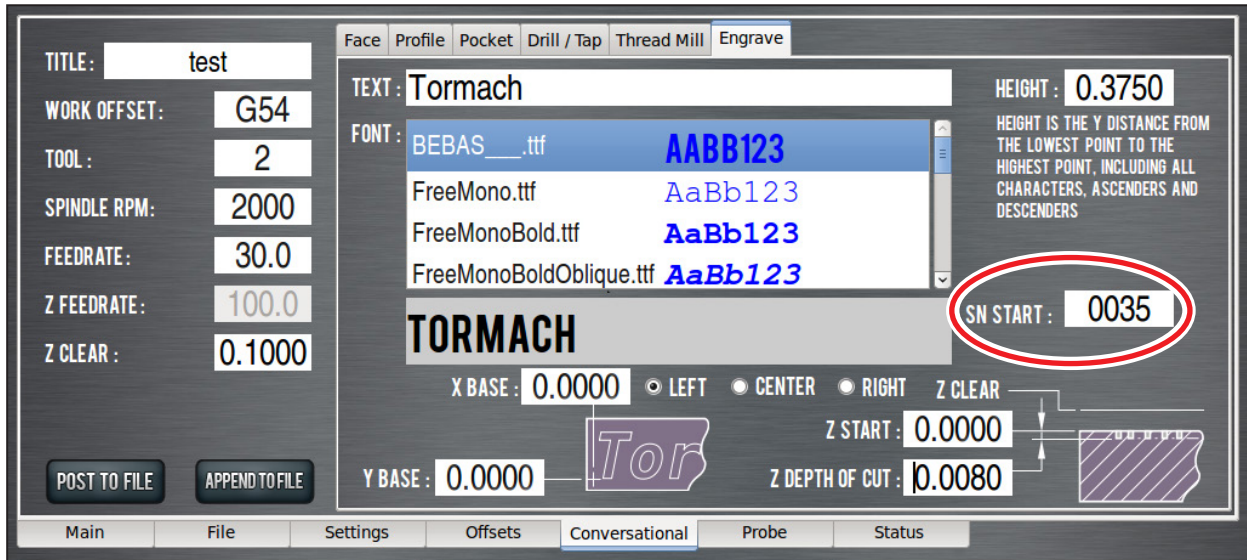


Figure 4

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.

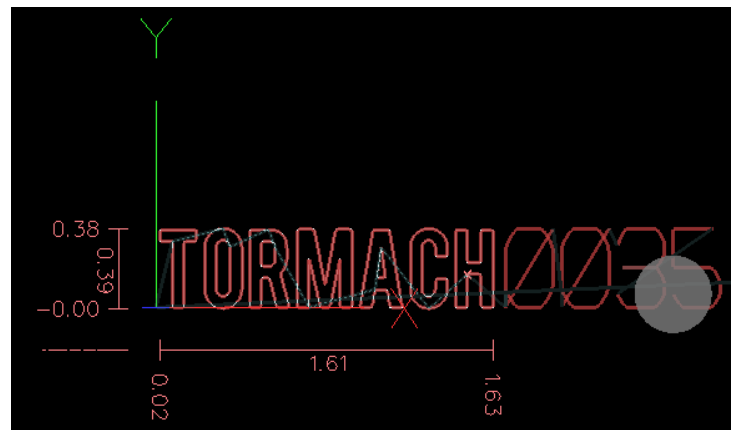


Figure 5

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
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**).

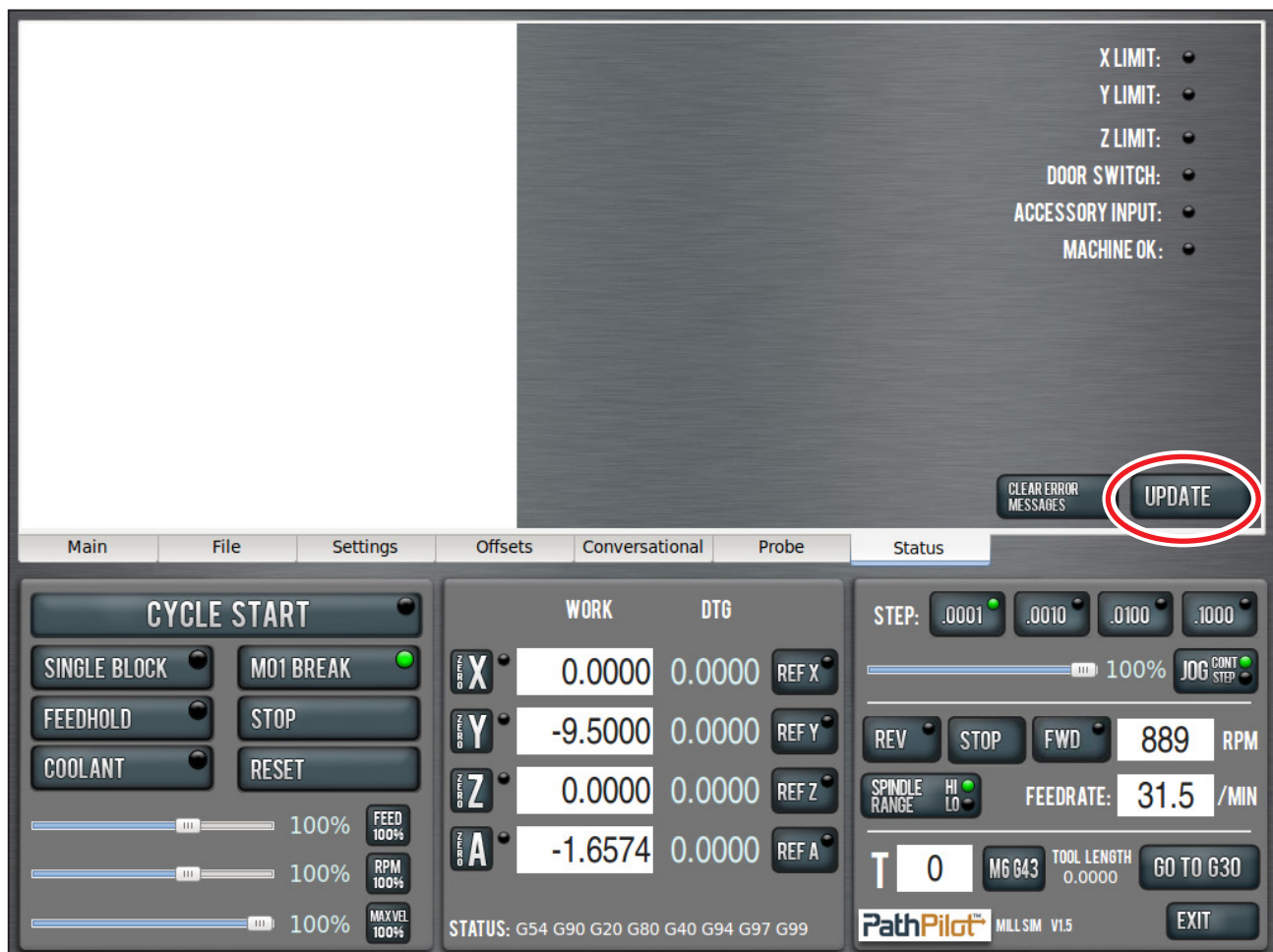


Figure 6

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**).

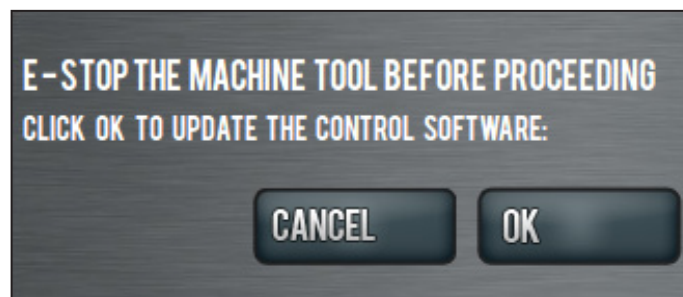


Figure 8