Release Notes for PathPilot™ v1.9.2

Release Date: September 1, 2015

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.*

Enhancements:

- #1006 Mill - Add ADMIN commands SET_X_LIMIT and SET_Y_LIMIT to set X and Y travel limits. Some machines have travel envelopes that are slightly larger than the specification, and this ADMIN command lets users set travel limits longer than the specification if their machine allows it. Example: ADMIN SET_X_LIMIT 18.25 sets max travel for X axis to 18.25 inches. Standard values for X and Y are 18.0 and -9.5. SET_Y_LIMIT must take a negative value. All values must be in inches.

- #991: Mill - Conversational enhancement: Option to use center of text as work X zero

Bug Fixes:

- #1026: Mill - Probing: Move and set A-axis center button has incorrect/misleading graphic

- #1025: Mill - Tool Table: Units revert to inches in G21 shortly after entering a value

- #1024: Mill/Lathe - ADMIN SETTINGS BACKUP fails to backup mill configuration files

- #1023: Mill/Lathe - G-code Editor: Allow editing G-code when machine is off

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

©Tormach® 2014. All rights reserved.
Specifications subject to change without notice.
SB0046_PathPilot_Release_Notes_0915A

Page 1

Release Notes for PathPilot™ v1.9.1

Release Date: August 20, 2015

Enhancements:

- #616: Mill - Add software support for door switch (hardware kit available shortly)
- #1008: Mill - Conversational: LH option for thread mill
- #991: Mill - Conversational enhancement: Option to use center of text as work X zero
- #806: Mill/Lathe - Allow export and import of tool table, fixture offsets, fonts, thread data

Bug Fixes:

- #1002: Mill - Tool Table: When putting in tool info, hitting return/enter between cells the focus is lost after about a second when you get to the "length" cell.
- #564: Mill - Interpreter: Spurious duplicate O word error message on M6 MDI command
- #1014: Mill - Conversational: Internal/External button on thread mill tab should clear the thread size drop down and the pitch and dia DROs
- #1015: Mill/Lathe - G code interpreter: DROs won't zero if a program is stopped between two G5x work offset commands and the second command is followed by a tool change, probing move, or M66 read input command. Workaround: reinstate G5x offset via MDI
- #1007: Mill/Lathe - Recent File Dropdown: Clear Current Program choice removes currently applied tool offset. Workaround: click M6/G43 button (mill) or re-enter tool number (lathe) to re-apply offset after CCP
- #1010: Mill - Conversational: Thread mill with 20 TPI generates a thread with 22 threads per inch.
- #982: Mill/Lathe - Tool lengths and diameters in Offsets tab display in inches units even when in G21 until tool information is changed
- #763: Mill/Lathe - ATC and USB IO: USB 3.0 blue ports do not work
- #971: Mill - Conversational Engraving: Zero RPM for use with drag engraver hangs program at first G1 move
- #977: Mill – Probing: Failure probing small diameter circular boss

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

©Tormach® 2014. All rights reserved.
Specifications subject to change without notice.
SB0046_PathPilot_Release_Notes_0915A
Page 2

Release Notes for PathPilot™ v1.9

Release Date: July 10, 2015

Enhancements:

- #767: Mill/Lathe - Filechooser enhancement: Should support details view (or small icons view)
- #940: Mill/Lathe - Homing enhancement: If machine has already been homed, pressing Ref buttons should result in OK/Cancel popup
- #916: Mill - Conversational - Thread Mill - take X/Y positions from drill table
- #955: Mill - Tool Change: Add tool description to tool change message (e.g. "Insert tool #4 (description) press Cycle Start to continue")
- #957: Lathe - Gcode interpreter: Support G74 canned drilling cycle

Bug Fixes:

- #765: Mill/Lathe - G code window: M30 should reset G-code listing to the top of the program.
- #971: Mill - Conversational: Allow zero RPM on engrave routine for use with drag engraver.
- #937: Mill/Lathe - DROs: Highlighting a portion of the field in the DRO will clear the un-highlighted data as well.
- #964: Mill - Interpreter: Stopping a program using the stop button at an M01 break before a work offset change will cause the G5x command after the M01 break to be interpreted. Ex: G54, M01, G55 - aborting the program at M01 results in G55 system being active.
- #967: Lathe – Machine-OK LED on Status tab does not indicate machine power state correctly.
- #936: Mill/Lathe - Cancel out of New Folder in File tab gives error.
- #970: Lathe – Conversational: Drill routine uses Tap peck depth DRO for peck depth until Drill/Tap button is clicked.
- #961: Mill/Lathe – Soft Keyboard: If user has soft keyboard enabled and types ADMIN CONFIG into the MDI line, the screen goes white and the config picker does not appear.
- #851: Mill/Lathe – Subroutines: External subs are not recognized unless they have an *.ngc* file extension. *NOTE: subroutines now accept .nc or .ngc, but not .tap, .txt, etc.*
- #969: Mill – Probing: Pressing *Stop* during a probing move results in the feed rate being set to a very small value.
- #926: Mill – Toolpath preview: XYZ graphic has green for X axis, red for Y. This is opposite of most CAD/CAM systems.
- #872: Mill/Lathe – File: Clear Current Program changes machine to inch units (G20) if it was previously in metric (G21).
- #962: Mill – A axis moves slowly and creeps the final fraction of a degree while DTG is already 0.0.
- #963: Mill – ETS: Move and set tool length button sets tool length correctly internally, but doesn't refresh the tool table display unless you page away from that screen and then return to it.
- #960: Lathe – Offsets: Tool geometry offset will *not* be applied when machine is first powered on. Workaround: after power on and reference, type a Tx command into MDI line to apply tool offsets.

- #931: Mill/Lathe – File tab: When a file is created with the *save as* drop-down menu choice of the G-code editor program it doesn't show in the folder until the folder is manually refreshed by the user.

Other:

IMPORTANT! Lathe: ESC keyboard shortcut for Stop button was removed due to conflict with MDI line defocus. Do not attempt to stop the lathe using the ESC key – use the spacebar (pause) instead.

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

©Tormach® 2014. All rights reserved.
Specifications subject to change without notice.
SB0046_PathPilot_Release_Notes_0915A
Page 4

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 the most recent PathPilot update file (vX.X) at www.tormach.com/pathpilot_tracking.html#history

2. Transfer this file to a USB drive.

3. Put the USB drive in your PathPilot controller.

4. With the PathPilot controller on and out of Reset mode, go to the Status screen and click the Update button (see **Figure 1**).
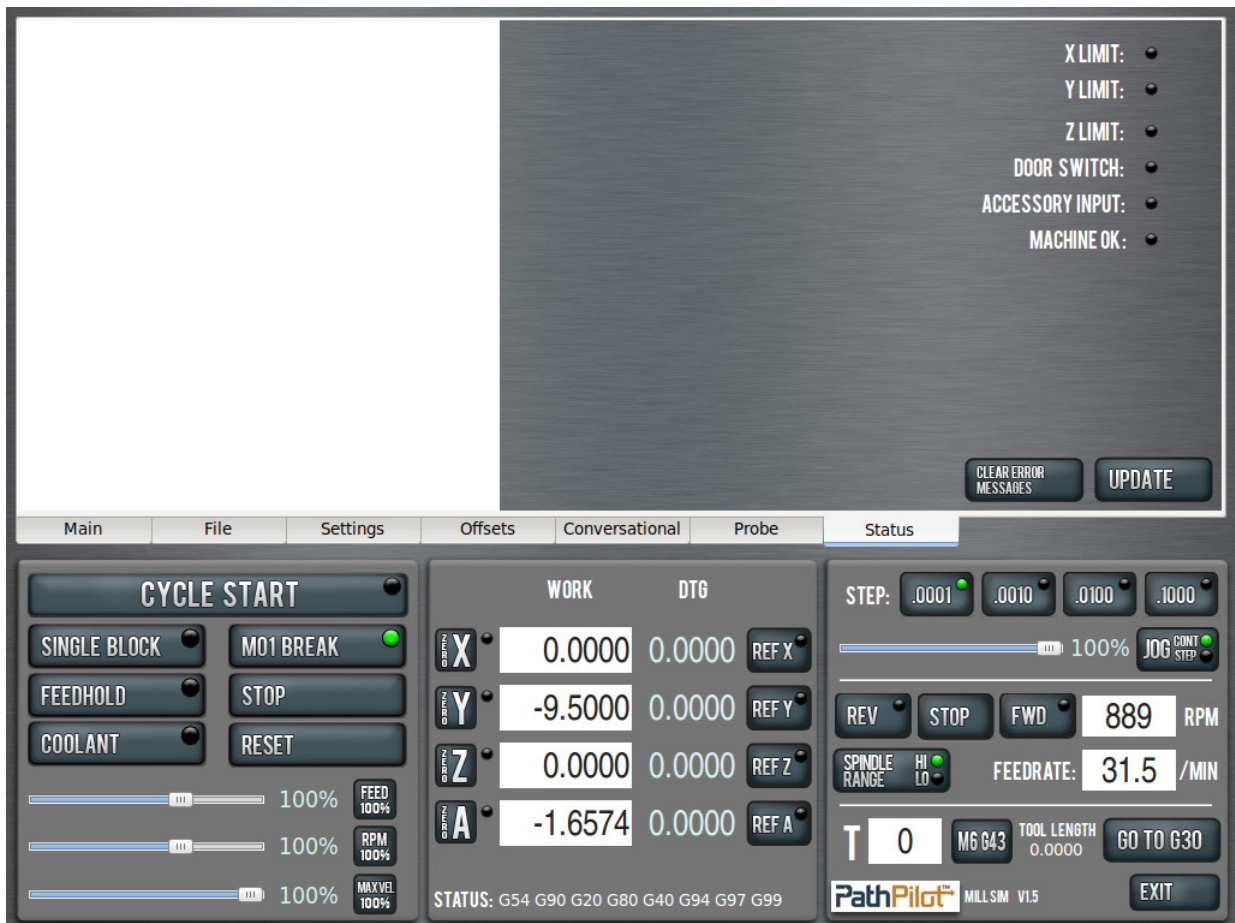

Figure 1

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

©Tormach® 2014. All rights reserved.
Specifications subject to change without notice.
SB0046_PathPilot_Release_Notes_0915A

Page 5

5. Navigate to the (vX.X) file on your USB drive and click Update (see **Figure 2**).
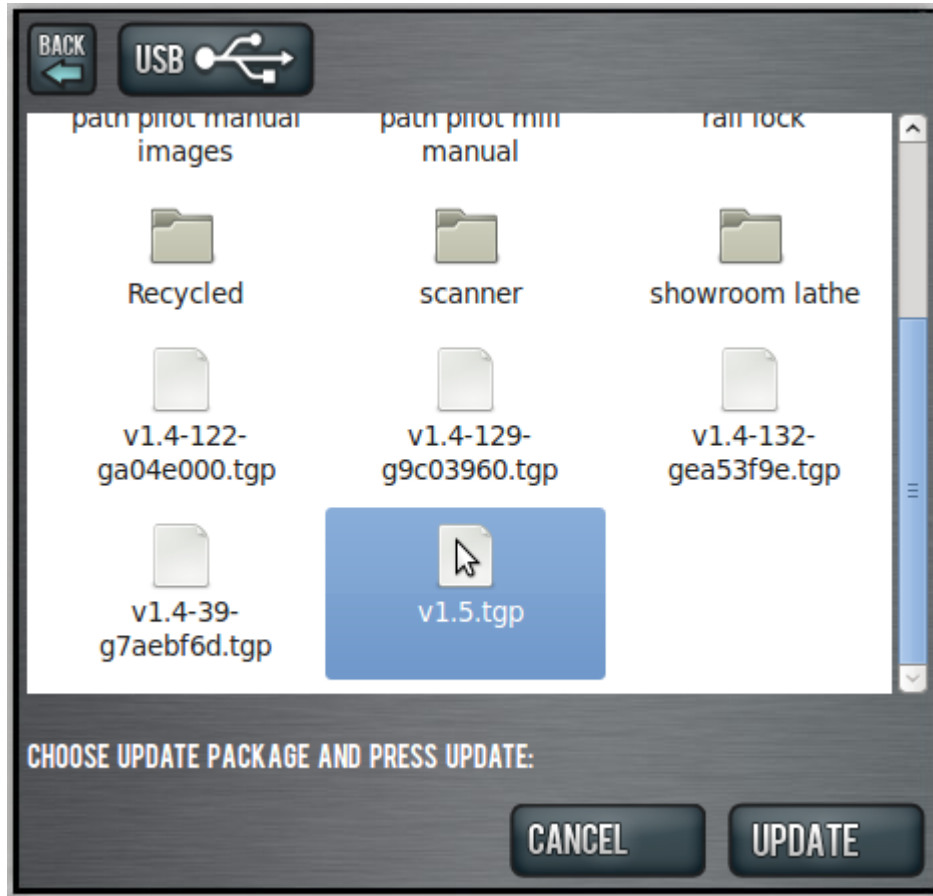

Figure 2

6. Push the E-stop button on the mill when prompted and click *OK* (see **Figure 3**).

Figure 3

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

©Tormach® 2014. All rights reserved.
Specifications subject to change without notice.
SB0046_PathPilot_Release_Notes_0915A
Page 6