TB175 (Rev1) - Rigid Tapping Troubleshooting

Symptoms	Possible Problem	Solution/Troubleshoot
Rigid tapping was working and now when running jobs the message "Option not availiable, line ??" appears.	Demo mode ran out and permanent unlocks not entered.	From the Main menu press **F7 - Utility**, then **F8 Options**. Verify Tapping Cycles and Rigid Tapping are on.
When running a G74 or G84 rigid tap cycle, the control stops above the hole to be tapped with a message in the status window "moving" and **DOES NOT** continue.	1. No spindle encoder feed back 2. No spindle encoder index pulse 3. On **CPU10** the 6th axis index pulse on CPU serial number 384 and before does not work. (**FOR SD1 only systems**)	1. Verify **parameter 35** is set to the correct axis (see TB123). 2 . Turn spindle by hand check for encoder movement (see figure 1), if no movement check spindle encoder, cable, and connections. Figure 2 shows the DB9 pin out. 3. To check for the index pulse on DC systems turn the spindle by hand and look for the '*' next to the axis for the spindle in the PID menu (see figure 1). 4. To check for the index pulse on AC systems set the bit to ignore the index pulse in **parameter 36** (see TB123). 5. Use the 5th axis encoder input on the CPU10 and change **parameter 35** to 20 (see TB123). 6. If steps 3,4 or 5 indicate the index pulse is missing check spindle encoder, cable, and connections. Figure 2 shows the DB9 pin out.
When running a G74 or G84 rigid tap cycle you get a "410: Z axis(3) position error"	1. Spindle encoder turning opposite direction of spindle motor. 2. Spindle speed too high for tapping 3. Wrong number of spindle encoder counts.	1.Change the sign on the value of **parameter 34** (see TB123) 2. Try tapping at a reduced spindle speed 3. See figure 1 below to check counts. When the spindle is turned 1 revolution the number of counts should change by the value of **parameter 34** (see TB123).
Tapping does not make it to the bottom of the hole. Not deep enough.	1. Not programmed deep enough. 2. Spindle is turning off too soon.	1. Check Intercon or G code program for depth 2. **Parameter 68,69 and/or 82** may need adjusted (see TB123)
Breaking taps in blind holes or tapping too deep.	1. Programmed too deep. 2. Spindle is turning off too late. 3. Spindle speed is too high	1. Check Intercon or G code program for depth 2. **Parameter 68,69 and/or 82** may need adjusted (see TB123) 3. Slow down spindle speed

Press **F1 - Setup**, **F3 - Config**, enter "PASSWORD", **F4 - PID** to see the menu below.

Figure 1: Encoder Input

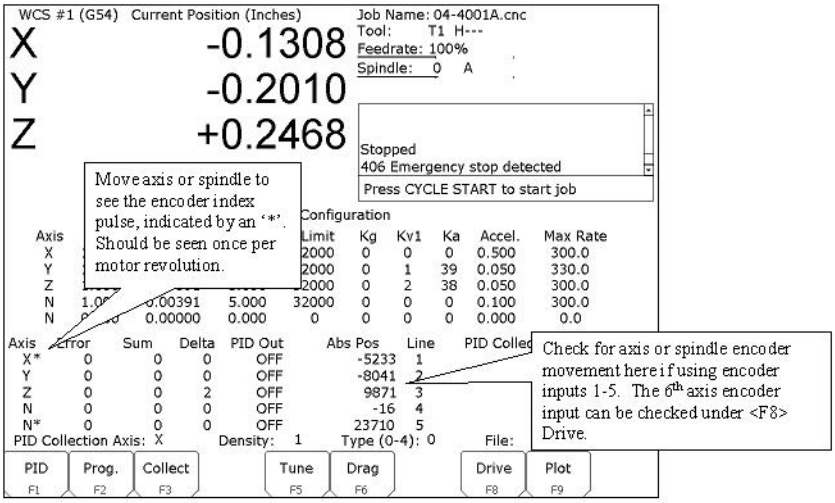
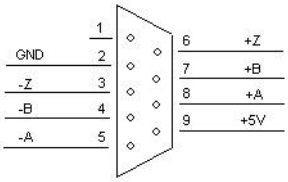


Figure 2: Spindle Encoder Pin Out



Document History

Rev1 Created on ***2005-02-04***