These technical notes describe the enhancements and/or corrections made to the software and are presented here with the most recent changes listed first. We welcome your comments and suggestions on the WEB at http://www.microkinetics.com/
****

TurnMaster Pro 2014 for Windows

Date: March 24, 2014
Subject: Enhancements resulting in revision 6.0.005
By: Jeff Kidd

1. Improved the speed of drawing the graphics of large CNC programs with numerous small circles and lines.

2. Corrected the labels of "*Start of Taper (X-Axis)*" and "*End of Taper (X-Axis)*" on the **Taper & Arc Roughing** screen, which were reversed.

3. Modified labels on the **Taper & Arc Roughing** screen to better reflect whether *Taper* or *Arc* was selected.

4. Enlarged the drawing area on the **Wizards | Threading** screen.

5. Added *Feed Rate Scale* to the **Machine Parameters** screen. All commanded feed rates in the G-code program will by multiplied by this number. The allowable values are between 0.1 and 100. The default is 1.

6. Corrected a problem when using a *Tool Turret* with multiple tools at the same station. Switching between these tools using the **M06** would sometimes give an erroneous error message.

7. Corrected the erroneous display of "*Invalid Tool Number ...*" during an **M06** command if the selected tool did not have a textual description.

8. Corrected a problem with the graphics after a tool change. Sometimes, the next tool radius was used when drawing the last move before a tool change.

9. When entering a *Drill* as one of the tools in the Tool Chest, assume the diameter is entered instead of the radius. For all non-drill tools, the radius is assumed.

10. Improved the graphics when using a *Drill* tool. Since a drill comes in from the side, it should not color a radius around the final tool position.

11. Modified the CNC display to highlight the 4th line (instead of the 3rd line) if the text size is small.

12. Corrected a problem with the **Diagnostics** process. Occasionally, the *Ramped Move* test would terminate before reaching maximum motor speed. This would result in a recommended *Ramped Move Speed* lower than it should have been.

13. Added "*Write Default VFD Settings*" button on **Machine Parameters** screen, **Serial Spindle** tab. This button enabled for the **Teco 7300** VFD only. When pressed, all default parameters stored in the file, **MK Lathe 1236.CV7** will be sent to the **Teco 7300** VFD.

    MK Lathe 1236.CV7 is now included in the installation package.

14. When using a Tool Turret, the turret position will be set to the (*Tool # mod 20*). For example, Tools 2, 22, 42, 62, and 82 will all use turret position #2, but can have different **X** and **Z** offsets. This feature enables multiple tools to be mounted at a single tool turret position.

15. Improved the wording on the **Teco 7300** VFD communication error message box to provide clearer and more useful information.

16. Improved the reliability of the **Get Settings** button on the **Machine Parameters** screen, **Serial Spindle** tab.

17. If a ***Teco 7300*** VFD is attached, but no **MN400** controller is connected, give the user the option to operate the VFD only using the ***Spindle*** button on the main form.

18. Corrected a problem with **Tool Name** descriptions, which were being converted to all lower-case.

19. Corrected a problem when a **G25** (Call Subroutine) command. If an invalid subroutine name was entered, the CNC program would be erroneously truncated.

20. Added "*Init as Start*" and "*Return to Start*" buttons on the MN400 Jog Screen. These buttons perform the same functions as the corresponding buttons on the Main Form's toolbar.

21. The installation process now adds the **RunCNC.exe** file which will be the default program for *.CNC* files if **TurnMaster Pro** and **MillMaster Pro** are both installed. This small program decides which program to launch (TurnMaster or Millmaster) if a .CNC file is double-clicked.

22. Added the read-only variable **TM_ToolNum**, which returns the currently active tool number (1-99).

23. Corrected a problem with **M25** (Return tool to start) in graphics only mode. If an **X** or **Z**-axis move was very small, then it would not update the lathe counter(s) correctly.

24. Added checkbox "*Prompt for Return to Start?*" on the **Machine Parameters** screen. The default value is checked. If this checkbox is unchecked then TurnMaster will no longer ask the user "*Return Tool to Start ?*" when running a program.

Date: May 29, 2013
Subject: Enhancements resulting in revision 5.2.155
By: Jeff Kidd

1. Corrected a problem on the **Wizards | Threading** screen. The generated **G01** codes on the *Sharp Edge Breaker Pass* (if selected) now correctly accounts for 0.050" clearance assumed in the generated **G33** (Threading) command.

2. The ***M06 - Tool Change*** message box is now larger and easier to read.

3. Added a ***Single-Step*** checkbox to the ***M06 - Tool Change*** message box. If checked, **TurnMaster** will enter Single-Step mode and stop before the execution of each line.

4. Corrected a problem with **G83** and **G87 Multi-Depth Cycles,** not working correctly when a **G92** command was active.

5. Added read-only variables **(TM_TOOLRADIUS)**, **(TM_TOOLXOFFSET)**, and **(TM_TOOLZOFFSET)** which will return the radius and offsets of the currently selected tool, respectively. See *Appendix F* in the user's manual for more information.

6. Improved the display screen for math syntax errors found in the G-code program.

7. Added Plunge Angle capability to the **Threading Wizard**. This option will generate a subroutine which will do a multi-depth thread, one level at a time with a variable *Plunge Angle* between 25 and 35 degrees.

8. Added **National Taper Pipe** (*NPT*) values to the threading table.

9. Corrected the problem where the main display redraws after switching to the full-screen editor when any change to the program was made.

10. Added <Cancel> button on the "*Save System Parameters?*" screen. If pressed, no action will be taken, and **TurnMaster** will not terminate.

11. Corrected a problem with the *MRU* (Most Recently Used) file list. Previously, if a file couldn't be found it was removed from the *MRU*. Now all *MRU* files are retained regardless of whether the file still exists.

Date: December 6, 2012
Subject: Enhancements resulting in revision 5.2.143
By: Jeff Kidd

1. Corrected a problem with **G83** and **G87** commands where the Z-parameter was used as an absolute position instead of actual depth. This was an isolated issue only on two recent releases issued in 2011. All prior versions and this release correctly treat the Z parameter as actual depth into the material with the standard .050” clearance.

2. Only write **.TIM** file when executing a G-Code program while in **Machine (or Both)** mode. Added the information for total running time, and if a program terminated abnormally.

3. The "z origin" value in the setup file (**.tps**) is now being saved as "Left" or "Right" instead of 1 or -1. Setup files using the old format continue to be supported.

4. Added entries to the **Threads.MDB** database to include additional standard threading sizes. With this release the Threading wizard is able to generate many more threads with a quick selection instead of entering custom value.

5. If a program is cancelled vie the **Key Pause** screen, then **TurnMaster** will restore the CNC display textbox to the line # that was being executed when the program was paused.

6. When in *Metric* mode, the F-parameter (Feedrate) and the D-parameter (Downfeed speed on multidepth canned cycle commands) are now interpreted as mm/Min. Previously these parameters were interpreted as (10 * Inches)/Min, regardless of Inches/Metric mode. No changes are made to the *Inch* mode.

Date: September 13, 2012
Subject: Enhancements resulting in revision 5.2.137
By: Jeff Kidd

1. Added a "**No pause on comment lines when stepping.**" checkbox on the *KeyPause* screen. The default value for this checkbox is *checked*.

2. If nothing changed on the **Options | Material Size** screen, then don't redraw main screen.

3. Corrected a problem on the **MN400 Jog Screen**. When using the RS-232 interface instead of USB, pressing a key on keyboard was not stopping the current jog in progress.

4. After completion of a program, write out a summary of tools used and respective machining times for each tool. The file will be named the same as the CNC program but with a **.TIM** extension. This appends new information to the existing file.

5. Save "Jog Distances" on the **MN400 Jog Screen** with up to 5 digits. Previously, these values were rounded to 4 digits when exiting and restoring the screen.

6. Allow distances equivalent to 1-step for the "Jog Key Resolution" values in the **Machine Parameters** screen, **Jog Keys** tab. Previously, the smallest value allowed was the distance equivalent of 2-steps.

7. Corrected a problem with **G33 Threading Cycle**. If the thread started at other than Z = 0, the spindle would not decelerate at the correct point in the move.

8. Corrected a problem with the graphics on a **G33 Threading Cycle** when a **G92 Set Tool Start Position** has been used. The threading graphic did not mirror correctly on the top and bottom of the X-axis.

9. Corrected a problem with the **G04 Dwell** command, which was not allowing a user keystroke to pause the program.

Date: June 19, 2012
Subject: Enhancements resulting in revision 5.2.130
By: Jeff Kidd

1. Added a delay value in the **Advanced** tab of the **Machine Parameters** screen. The default value is 0.003 seconds and is needed to prevent fast PC's from causing a communication error with the MN400 which caused a hang up.

2. Corrected a problem with **M25** (Return to Home Position) in graphics mode. Under certain conditions, the graphical tool's X-position would display incorrectly.

3. Improved the labeling of the visual grid. With some material sizes, some Z-grid labels were missing as they attempted to display at the top and bottom edges of the screen.

4. Previously, the spindle stopped anytime a tool change is commanded. This has been corrected such that, during a tool change, it stops the Spindle and Coolant only if the *Tool Change Pause* checkbox is checked. This bug was present in Ver 5.2.125 released in Spring 2012.

Date: May 10, 2012
Subject: Enhancements resulting in revision 5.2.125
By: Jeff Kidd

1. In the **Machine Parameters** screen, added the ability *Save As* a filename in a different directory. Previously, *Save As* only worked on the current working directory

2. Corrected a problem in the **Tool Chest** screen. If a different tool was selected while in Machine mode and the MN400 Controller was turned off, a ***Retry / Continue Offline*** message box appeared. Selecting **Retry** was causing an "Error 5" message and TurnMaster would terminate.

3. Allow "Single-Step with Canned Cycles?" checkbox on the *Program Pause* screen to work in Graphics Only mode.

4. On the **MN400 Jog Screen**, if using an *Axis-3 Stepper Motor*, , pressing the *Advance Tool Turret* button now turns on **Output #8**. This is now consistent with the tool change command as described in Item #8 of the release notes for Version 5.2.114.

5. Added a checkbox "*Write values to part program?*" to the **Options | Tool Start Positions** screen. If checked, the X and z start positions will be written, as metacommands, to the part program.

6. Added ability to modify the order of axis movement during an M25 command. The default order for a standard tool is **X** then **Z**. If the current tool is a *rotated* tool, the default order is **Z** then **X**.

```
For example: M25 XZ     \-- Move X then Z-axes.
             M25 z      \-- Always move Z-axis first.
             M25        \-- Use default order of axes.
```

7. Added *read-only* access to Tool Radii and Offsets. Use the system variable names **TM_TOOLRADIUS***x*, **TM_TOOLXOFFSET***x*, and **TM_TOOLZOFFSET***x*, where *x* is a valid tool number from 1 to 99. See **Appendix F** in the User's Manual for a complete list of system variables.

8. Added the ability to view values of *user-defined* or *system variables* (variables within parenthesis "( )") or *evaluated formulas* (mathematical formulas between bracket pairs "[ ]") via the comment line of an **M00** command.

For example: G28 (NextYear) = [YEAR[0] + 1] /Set Value
M00 / Next Year = (NextYear)

Or simply:
M00 / Next Year = [YEAR[0] + 1]

Would display the M00 Message Box with the message:

"Next Year = 2013"

9. Added a new **M01** command. **M01** is identical to **M00** with the exception that all *Coolant* and *Spindle* outputs (Outputs 1, 2, and 5), and any *VFD Spindle Controllers* (Teco 7300, or Fluxmaster 100) will be turned off while the message box is displayed, and restored to their original state upon completion of the command.

Date: February 28, 2012
Subject: Enhancements resulting in revision 5.2.114
By: Jeff Kidd

1. If a Tool Changer (*Axis 3 Stepper* or *AC Motor*) is selected in the **Machine Parameters** screen, the *Tool Change Pause* checkbox will be unchecked and disabled.

2. Removed the "Serial Spindle" checkbox on the **Machine Parameters** screen, **Serial Spindle** tab. Now, the user simply selects among three radio buttons (*None*, *Teco 7300*, and *FM100*).

3. When *Feed Hold* mode is turned off, and the safety shield is opened, stop rotation of the spindle in addition to stopping X, and Z movement. The spindle is restarted if the <Continue> button is pressed on the *Pause* screen.

4. Modified the **G81** command using the ***i*** (depth per pass X-axis). Previously, the i-parameter was used to determine the number of passes, and the depth would be divided equally between passes. Now the actual depth specified in the **i**-parameter is used, with any leftover amount cut during the last pass.

5. Modified the **G82** command, as specified above for the **G81** command. In addition, the ***i***-parameter was changed to ***k***, to better represent that movement is along the z-axis.

6. Corrected a problem with **Control | Reposition From Limits** function. If the current tool was not Tool #1 and had different **X** and/or **Z** offsets from Tool #1, then the tool would not return to the home position correctly.

7. Corrected a problem that occurred when a G-Code program is running past midnight. Under certain rare conditions, **TurnMaster** would not handle the date change correctly and hang.

8. During a Tool Change, if using an *Axis-3 Stepper Motor*, the software now turns on **Output #8**. With the proper add on circuit, this disables the **FeedRate Override** on the MN400 during tool changes and provides very reliable high speed tool changes.

9. Added r*ead-only* access to most system variables in the **Machine Parameters** screen. See the new **Appendix F** in the User's Manual for a complete list of variables.

Date: November 30, 2011
Subject: Enhancements resulting in revision 5.2.108
By: Jeff Kidd

1. After jogging the tool via the MN400 Jog Screen, if in **Both** (Machine and Graphics) mode, the graphical tool image position was not being updated.

2. When issuing a Return to Start command (via the menu, or M25), if the X-axis is greater than the X-home position, move the Z-axis first, then the X-axis. Otherwise, move the X-axis first. Previously the X-axis was always moved first regardless of the current tool position.

3. Corrected a problem on the *Spindle Control* screen used with the FluxMaster 100 or Teco 7300 spindle controller. If a spindle RPM greater than 32,767 was entered, an error occurred and TurnMaster would shut down. That error now is handled with an error message.

4. Added a "Single-Step with Canned Cycles?" checkbox on the *Program Pause* screen. If checked, there will be a pause between individual moves within each canned cycle.

5. Modified the *Control | Reposition From Limits* and *Control | Reprogram Limit Censors* functions such that after homing to the limits, the Z-axis returns to home followed by the X-axis. Previously, the X-axis returned to its home position first.

6. Corrected a problem with the baud rate when using an MN400 with a serial connection. Baud rates other than 9600 were being ignored in the **Machine Parameters** screen.

Date: October 14, 2011
Subject: Enhancements resulting in revision 5.2.103
By: Jeff Kidd

1. In the **Tool Chest** screen the "Side Tool" (i.e. Boring Bar, Drill, etc.) checkbox has been removed. Now, the dimensions of newly selected image file are automatically detected and the orientation is set appropriately.

2. If using a "Side Tool" (i.e. Boring Bar, Drill, etc.), the **M25** (Return Tool To Start) will retract the Z-axis first, followed by the X-axis. For conventional tools, the existing functionality, of retracting the X-axis first, is retained.

3. Added a warning message if user tries to change the **X** and/or **Z** tool offsets in the **Tool Chest** screen.

4. On the **MN400 Jog Screen**, improved the error message to give more information, when trying to touch off a tool other than Tool #1.

5. On the **MN400 Jog Screen**, if any **X** and/or **Z** tool offsets were changed, added a message box asking if the user wants to write these values as Meta-Commands to the current part program.

6. In *Windows Explorer*, a new feature has been added, such that if the user double-clicks on a **.CNC** file, and **TurnMaster** is already loaded, the existing instance of **TurnMaster** will try to load the selected file. If **TurnMaster** is busy, it will not try to load the selected file.

   Note: The **TurnMaster** application must be associated with the **.CNC** extension via *Windows Explorer* for this to work.

7. Corrected a problem when the first line of a program contained a MetaCommand containing a comma ",". In this case the comma was not being parsed correctly.

Date: August 31, 2011
Subject: Enhancements resulting in revision 5.2.100
By: Jeff Kidd

1. Added an optional **D** (Dwell) parameter to the *G33 Canned Threading Cycle* command. If a D-parameter is seen, TurnMaster will dwell for the specified number of seconds after each threading pass.

2. Added menu options **Edit | Enlarge Font** and **Edit | Reduce Font**. The default font size is 8.25 pt and can be enlarged up to 16 pt.

3. Reduced the RPM lower limit for the **G96** (Constant Surface Speed) command from 60 RPM to 5 RPM. This is the lowest RPM that will be commanded by **TurnMaster** during *Constant Surface Speed* operation.

4. Improved the error handling on the **Teco 7300** spindle controller. If a speed greater than the maximum allowable speed is issued, **TurnMaster** was waiting unnecessarily before executing the next G-Code command.

5. Improved the operation of **G96** (Constant Surface Speed). The minimum speed now allowed during CSS operations is now 5 RPM (previously it was 60 RPM). Also, **TurnMaster** now initializes the spindle speed as soon as a **G96** is seen, instead of waiting for the first X-axis move. The RPM display, during CSS operations, has also been improved.

6. Added "Reverse Lock Steps" input field on **Machine Parameters** screen, **Advanced** tab. This field is used for tool changers attached to an axis-3 stepper motor, with reverse locking.

7. Added 2 buttons, "*Advance Tool Turret*" and "*Tool Turret Rev Lock*", on the **MN400 Jog Screen.** These buttons are only used with axis-3 stepper tool changers.

8. Added a new frame "*Axis-3 Tool Turret*" and 2 buttons "*Advance*" and "*Rev Lock*". These buttons are only used with axis-3 stepper tool changers.

9. When an **M00** command is processed, display the comment (if any) in the message box. This allows the user to customize the message.

10. Corrected a problem processing the *{UNITS}:* metacode. If the specified unit (Inches or Millimeters) changed, not all the conversions were being done.

11. Convert the "Jog Distances" when converting units. Previously, these values were not being converted.

12. Disabled line autowrap from the editor Display. Previously, a line wrapped, the displayed line # would not match the actual line # when the program executed.

13. Added part origin parameters to **Options | Material Size** screen. These values may now be independently attached to the current program or the default setup file, or both.

14. Added "*Used*" checkbox for each tool, and a button to "*Write Tool Meta-Commands to Program*" to the **Tool Chest** screen. Clicking this button will write a "{TOOL}:" meta-command, in the current G-Code program, for each tool whose "*Used*" checkbox is checked.

15. Added a message box, reporting estimated machining time, at the end of a program run in *Graphics* mode.

16. If the user tries to display the Help file and Adobe Reader is not found, then display a reminder to the user to install Adobe Reader.

Date: June 2, 2011
Subject: Enhancements resulting in revision 5.2.75
By: Jeff Kidd

1. The file I/O now retains the current directory after saving or loading a file in various ways. Previously if **TurnMaster Pro** was loaded by dragging a file or double clicking a file, then the source directory was not remembered.

2. Display thumbnails on the left of the PDF Help File when initially displayed. All other PDF options are retained from the previous viewing of the help file.

3. Corrected a problem in *Millimeter* mode, when some arcs were reporting an incorrect **I** and/or **K** parameters. The test for error was adjusted to take into account the millimeter vs inch mode and to use a proper tolerance.

4. **Absolute Programming Mode** (G90) is now the default for **TurnMaster** if no G90 or G91 command is present. Previously **Incremental Mode** (G91) was the default. If a G90 or G91 is issued at the start of the program (recommended), then the program will work as it previously did.

5. **Multiple Arc Programming Mode** (G75) is now the default mode for **TurnMaster.** Previously **Single Arc Programming Mode** (G74) was the default mode. If running an older CNC program a G74 may be added at the beginning to work as did previously.

6. Corrected a problem where the screen was unnecessarily being redrawn when running a CNC program that contained Meta Commands.

7. For consistency between programs, **Edit | Undo** is now the first selection listed under the Edit menu.

8. Improved the **File | DXF Import** function as follows:
   - Coordinates are written with up to 5 decimal places.
   - Extra comments are written to the output file.

9. The default Feed Rate on the **MN400 Jog Screen** is changed from 10 ipm to 5 ipm.

10. Corrected a problem when selecting *Top Side* Tool Image Location in the Machine Parameters screen. The tool was not always positioned properly.

Date: March 22, 2011
Subject: Enhancements resulting in revision 5.2.67
By: Jeff Kidd

1. Do not process the *Pause Program Execution* functions (Key_Press event on the Main Form) while the Tool Changer is active.

2. Corrected a problem when drawing the yellow material on the screen. Sometimes the yellow color would "spill over" the edge of the material by an amount equal to the radius of the last tool used.

3. Enhanced the **M03** and **M04** commands when using the FluxMaster FM100 or Teco 7300 serial spindle controller. The S-parameter (spindle speed) can now have a negative value. This will have the effect of not waiting for the spindle to spool up before continuing to the next G-code statement.

4. Added **G64** - *Turn ON Continuous Contouring* and **G60** - *Turn OFF Continuous Contouring.* The checkbox on the Machine Parameters screen will be the initial setting of *Continuous Contouring* each time a program starts up.

5. Corrected a problem when certain math operations resulting in a large number of decimal places. A dubious error message "*Wrong sign or bracked placement*." was being generated.

6. Improved the drawing of the grid. When the number of units was very large, too many grid lines were being drawn.

7. When jogging and the system is in Metric mode, the large distance warning for fixed moves was set at 10mm. It is now set at 254mm or the equivalent of 10 inches.

8. Corrected a problem when opening a .CNC file from the Windows File Folder, **TurnMaster Pro** would not always auto-load the file selected.

9. Changed the acceptable input range of the "*Taper Angle*" field on the **Threading Wizard** screen to -45 to 45 degrees. Previously the range was 0 to 90 degrees.

Date: December 16, 2010
Subject: Enhancements resulting in revision 5.0.52
By: Jeff Kidd

1. Added an option in the **Machine Parameters, Advanced Tab** to allow curves to be approximated by multiple chords.

2. Added # of Chords per Circle to the Advanced tab of the Machine Parameters screen. Previously, **TurnMaster** always used 360 chords per circle when using chords.

3. Added **M94 - Issue Literal Command to MN400 Controller** functionality. Using this M-command, you can send a native MN400 command to the controller unedited. The command can be surrounded by double quotes, but this is not required. This command will be ignored if not in Machine mode or a different controller is used.

   Examples: **M94 m1000,500**
   m94 "m1000,500"
   m94 m1000,500 /MN400 native move

   Any of the preceding commands will issue a native command to the MN400 Controller, moving the X-axis 1000 steps and the Z-axis 500 steps.

4. Corrected a problem in the **Tool Chest** screen. Under certain conditions, the *Side Tool* checkbox was not being saved correctly.

5. Added **G83** and **G87** Canned Drilling Cycles functionality. See the documentation for a detailed explanation of these functions.

6. Corrected a problem in the **G33** (threading) function. Previously there may have been a small delay at the end of the threading cycle before retracting the tool. This would cause an unwanted groove at the end of the thread.

7. Corrected the tab order on the **Material Size** screen.

8. Corrected a problem with certain arcs in the **Taper & Arc Roughing** screen. The Z-values of the roughing cuts were sometimes invalid.

9. Corrected a problem with an unnamed and unsaved program. Sometimes changes were not saved correctly when pressing **<F5> Auto Run**.

10. Added *Extend Rough Cuts to: (Z-Axis)* field in the **Taper & Arc Roughing** screen. When backing out of a taper or arc rough cut, continue to this new Z-coordinate. Previously, the Z-axis would back out only to the starting Z-position of the taper or arc.

11. Do not report an error when closing **TurnMaster** if the MN400 controller was turned off before exiting.

12. Corrected a problem saving the parameter *Origin Left*.

13. When closing the application using the **X** in the upper left corner of the main screen, perform the same end-of-program checks as when closing using the **File | Exit** menu selection.

14. After running diagnostics, the *Diagnostic Results* screen defaults to **No**, when asking whether to save the results.

15. On the **Diagnostics Results** screen, display *N/A* in the results summary for any axes not tested.

16. Disable entry fields for *Unramped Distance* and *Ramped Distance* for any axes not selected.

17. Added *Electric Pulse Tool Turret* in the **Machine Parameters** screen, *Advanced* tab.

18. Corrected a problem with the Math Evaluator. If several very small numbers were added (subtracted) there was a possibility of an error accumulation due to rounding.

19. Graphically draw internal threading using thin dashed lines. Previously, only external tapers were drawn in graphics mode.

20. Corrected a problem when executing a **G02** or **G03** (Circular Cut) after an **M97** (Wait for Input). If you used the **R** (Radius) parameter you would sometimes get an error stating the arc endpoints or center points were not correct.

21. Don't redraw the **CNC Program Window** after saving the file. This was resulting in the loss of the cursor position within the program.

Date: April 20, 2010
Subject: Enhancements resulting in revision 5.0.31
By: Jeff Kidd

1. Added the option for arc roughing on the **Taper** wizard and renamed it to **Taper & Arc Roughing** wizard.

2. Corrected computation of ***Start of Taper (X-Axis)*** and ***End of Taper (X-Axis)*** on the **Taper & Arc Roughing** wizard.

3. Improved the looks of the displayed arrowheads on the **Taper & Arc Roughing** wizard.

4. The Angle of Taper is now displayed in the range -90º to 90º on the **Taper & Arc Roughing** wizard.

5. Improve the **MN400 Jog Screen** to reestablish the correct jog speed after reconnecting to the controller.

6. Corrected a problem with **File | Save**. On a newly created file, after the initial save, subsequent saves without an intermediate ***Open*** would not save the changes.

7. Display the currently active tool instead of Tool #1 when initially displaying the ***Tool Chest*** screen.

8. Do not redraw the screen when exiting the ***Tool Chest*** screen.

9. Corrected a problem when the ***Run From 1st Line*** menu item is not checked and a VFD spindle controller is installed. If the user selected **<Cancel>,** when asked to enter the line # to start from, the Automatic Spindle warning message would display erroneously.

10. Added a ***Always Run From First Line*** button on the ***Run From Line #*** screen. When pressed the ***Run From 1st Line*** menu item will be checked and all subsequent runs will be from the first line.

11. On the **Find/Replace** text edit screen, the ***Replace Next*** button was changed to ***Replace This***. Pressing this button now changes the currently highlighted text to the replacement text.

12. Display a message reminding the user to restart ***TurnMaster*** if the backlash values have changed in the **Machine Parameters** screen.

13. Clear any old filenames when performing **File | New**. Previously the last open filename was retained if a subsequent **File | Open** was performed.

14. If no changes were made to the *Serial Spindle Control* parameters, then don't attempt to reinitialize spindle controller when exiting the ***Machine Parameters*** screen.

15. Allow user option to skip retrying of the connection to the *Teco 7300 Serial Spindle Controller* for the current session.

16. Improved the accuracy of ARC endpoint calculation on the **File | Import .DXF** function.

17. Changed product name to **TurnMaster™ 2010**.

18. Added up to 9 most recently used (MRU) files on the **File** menu.

19. Improved the performance of threading. In previous versions, **TurnMaster** would delay several seconds between threading passes.

20. Remove focus from the ***Spindle*** and ***Coolant*** buttons, in the **MN400 Jog Screen**, after they are pressed. This prevents an accidental pressing of the button during motor motion if the <*Space Bar*> is pressed to stop motion.

21. Display the updated the position counters after a ***Tool Change*** if in machine mode. Previously, the updated position values were not displayed until a subsequent move occurred.

22. When opening an existing file, the Line # on the status bar is now set to 1 instead of 0.

23. Corrected a problem when selecting File | Open. An erroneous dashed rectangle was sometimes being drawn.

24. Corrected a problem with the Math Evaluator. If a number immediately following an open bracket "**[**" was entered without any decimals to the left of the decimal point (ie .123) then an error occurred. For example the line "*G01 x[.95+(LastX)]*" was reporting an error, but the equivalent line "*G01 x[0.95+(LastX)]*" worked correctly.

25. Added menu option **Edit | Undo**. This menu selection is only available in full screen edit mode, and reproduces the functionality of the ***<Ctrl> Z*** hotkey.

26. Corrected a problem when saving a newly created file. Occasionally, the **File | Save** menu selection would ask for a filename even though the file has already been saved.

Date: October 14, 2009
Subject: Enhancements resulting in revision 5.0.8
By: Jeff Kidd

1. Corrected "Error 6: Overflow" when machining arcs if a signal was received from Input #8.

2. Modified the **Edit | Find/Replace** screen to easily toggle between Find and Replace modes.

3. The *Replace All* button on the **Edit | Find/Replace** now replaces all occurrences regardless of the cursor position.

4. Enhance the *End of Program* message box to include total elapsed time, time machining, and time waiting for user input.

5. Corrected a problem on Vista machines. When opening a new file, the display would sometimes redraw unexpectedly with the wrong zoom values.

6. Corrected a problem on Vista PCs only. When opening a new .CNC file, the graphics window would unexpectedly zoom-in before asking "*Restore Graphics ?*".

Date: September 23, 2009
Subject: Enhancements resulting in revision 5.0.6
By: Jeff Kidd

1. On the Machine Parameters screen, the name of the "Closed Loop" tab changed to "Backlash" to better describe entry fields.

2. When running a G-code program, changed the color of the currently highlighted G-code line (the line currently executing). Previously, the current line was difficult to see.

3. Added "Axis 3 Turret Speed" entry to the ***Machine Parameters, Advanced*** tab. If an tool turret is installed on axis 3, this will be the speed it rotates (steps/sec) when an **M06** tool change is commanded.

4. Keep track of accumulated partial step errors during **Jog**, **Move to Point**, and **Interactive Commands.**

5. On the Tool Change message box, details of the next tool are displayed.

6. Corrected a problem in the **MN400 Jog Screen**. If the jog speed was very slow (< 0.5 inches/min), the screen counters would sometimes stop prematurely.

7. Corrected a problem with the *Show Last* button on the **Diagnostics** form. All *comma (",")* characters were being interpreted as *carriage return, line feed (<CR><LF>)* characters.

8. On the **Diagnostics** screen, display a brief summary of the results at the bottom of the scrollable textbox.

9. Added *Loop* counter to the status display above the CNC code window.

10. Added new Toolbar items for *Return Tool To Start*, and *Init As Start Position.*

Date: April 23, 2009
Subject: Enhancements resulting in revision 5.0.3
By: Jeff Kidd

1. Corrected a problem with **G92** (Set current program position). If multiple **G92**s were present in a program, the shift values would not always be correct.

2. Corrected a problem with **Control | Return Tool to Start**. If a G92 was used in a G-Code program, the first **Return to Start** issued would not clear out the shift values.

3. Corrected a problem with the ***MK_Install.EXE*** program shipped on the installation CD, that affected **Windows 98** PCs only. The program would display, but the buttons did not function.

4. Corrected a problem with **G02/G03** arc commands using the R (radius) parameter. Occasionally, a rounding error would cause an incorrect error message.

Date: April 14, 2009
Subject: Enhancements resulting in revision 5.0.2
By: Jeff Kidd

1. Corrected a problem with the Keypad/Display on the **Teco7300** VFD. If the Gear Selection on the ***Options | Machine Parameters, Serial Spindle*** tab changed, the readout on the **Teco7300** would not always display correctly.

   Note: The spindle was spinning at the correct RPM. Only the display was incorrect.

2. Modified the install package to set the **MultiCAM** directory to be Read/Write for all users.

3. Corrected a problem with **File | Save As**. Sometimes on the first attempt to save the default directory was not set.

5. Added "**Sel**" (Selected Tool) option buttons to the **Tool Chest** screen. Clicking this button next to a tool, then pressing <Save> or <Accecpt> will have the result equivalent to executing a **"M06 T*t*"** CNC command.

6. Set the initial focus on the **Machine Parameters Password** screen to the password entry field.

7. When selecting File | New, **TurnMaster** no longer displays the message box *"System Parameters have changed...".*

8. If the "*Skip checking for a controller connection for this session.*" checkbox was selected on the **MN400 Communication** screen, then no longer check for a Serial Spindle connection either.

9. Corrected a problem that occurs when using a Serial Spindle Controller. If the user selected *"Cancel"* to the *"Automatic Spindle Controller is active... Would you like to continue?"* message, then the CNC Edit Window would revert to the last saved copy the CNC file, losing any changes since the last save.

10. Corrected a problem in the MN400 Jog Screen. If the radio button for a specific *Jog Distance* was selected and the associated distance was blank, the user would receive a "Type Mismatch" error and TurnMaster would terminate.

11. Added **Coolant On/Off**, and **Spindle On/Off** buttons to the ***MN400 Jog***, and the ***Move to Point*** screens.

12. If **Control | Reprogram Limit Sensors** was executed, TurnMaster will remind the user to "Save Parameters?" before exiting.

13. Corrected a problem after running a program in Graphics mode, then zooming an area using the mouse. The rectangle box drawn was sometimes a thick solid line instead of a thin dashed line.

14. TurnMaster no longer redraws the grid after running a program in Graphics mode. Previously, this could obscure tool path lines just drawn.

15. In the **File | Open** dialog box, added ***All Files (*.*)*** as an allowable file type in addition to ***CNC Files (*.CNC)***.

16. Added a checkbox to *"Save values to setup file."* on the **Options | Material Size** screen to allow the new values to be immediately written to the setup file.

17. If the **Material Size** changed, but the values were not saved, then remind the user that *"System Parameters have changed."* before exiting TurnMaster.

18. Added menu option **View | Redraw Screen *(<Ctrl> D)*** which will clear a previous graphical run, but retain the viewport size and location.

    Note: The **View | Restore Screen *(<Ctrl> R)*** menu option still clears the graphics and restores the location and viewport size to their default values.

19. Improved the function of **Zoom In** and **Zoom Out** buttons. Zooming now maintains the center point of the current view.

20. The **Pan** button now toggles Pan mode **On** and **Off**. Before this release, pressing the **Pan** button turned Pan Mode **On**, regardless of its current state. The only way to exit Pan mode, was to press one of the three Zoom buttons.

21. Added a **<Cancel>** button to the ***M00 Temporary Stop*** message box. Now, the user has the opportunity to press **<Cancel> to terminate** the part program.

22. Added a warning message on the **MN400 Jog Screen**. If the user enters a measured move of at least 10, with no decimal point, then warn the user of a potential entry error.

23. Corrected a problem when double-clicking on the full screen edit window would sometimes re-zoom the resulting graphics screen.

24. Corrected a problem with the graphics associated with drawing a thread. Occasionally the graphics would draw the thread outside of the actual threading area, or draw a stray line outside the threading area.

    Note: This only affected the graphical image on the screen and
    not the actual machining of the thread.

25. Increased the size of the CNC Display window to accommodate more lines of the CNC program to be displayed.

26. Added an icon on the top toolbar to toggle the CNC Full-Screen Edit mode On/Off. Previously, double-clicking in the CNC Edit Window would toggle the Full-Screen mode, but this feature has been disabled. Double-clicking in the CNC Edit window now simply highlights the current word.

27. Moved ***Start Position*** and ***Tool Chest*** menu items from the **Tools Menu** to the **Options menu**. Renamed the **Tools Menu** to **Wizards Menu**. These changes should help to avoid confusion between the old **Tools Menu** and the ***Tool Chest***.

28. Corrected a problem after saving a file, the cursor position in the CNC Display textbox would move to the beginning of the file. Now the cursor will retain its position after the save is completed.

29. Added support for the **R** (Radius) Parameter in the **G02/G03** commands. If the **I** and **K** parameters are not specified and the **R** parameter is specified, **TurnMaster** computes up to 2 possible arc center points and draws the shortest of the 2 possible arcs. If the **R** parameter is negative the longer of the 2 possible arcs is drawn.

30. Corrected a problem introduced in Version 4.3.74 where the **Continuous Contouring** and **Track External Control Panel** options, in the Machine Parameters screen, were being inadvertently cleared out.

31. Added two checkable menu selections under the **View** menu. **Thin Path Lines** and **Arrows on Path Lines** will change the way the graphics displays in order to allow the user to see more detail in the cutting of the part.

32. Corrected a problem when running a program in Graphics mode, the **Line** # displayed was not always updating correctly.

33. Added a reminder message to save system parameters after executing ***Control | Reprogram from Limit Sensors***.

34. Corrected a problem when touching off tools from the **MN400 Jog Screen**. If some tool other than Tool #1 was active, the touch-off values saved by **TurnMaster** would be relative to that tool instead of Tool #1. **TurnMaster** will now prompt the user to change the active tool and touching off Tool #1 before touching off any other tool.

35. Added a one-time "Terms & Conditions" notice.

36. Display an "Controller Offline" message if user tries to perform a function that requires connection to the controller, and it is Offline.

37. Previously ***Control | Reposition from Limits, Control | Reprogram Limit Sensors*** and ***Control | Home to Limits***. If the user aborted a move in progress, the CNC Edit Window would revert to the last saved copy the CNC file, losing any changes since the last save. This has been corrected.

38. Improved the performance of the Teco7300 VFD when changing directions. Previously, there was an extra 10 second delay when changing directions from M3 to M4 without using M5 to stop the spindle.

39. Corrected a problem when opening a **.CNC** program while in ***Both*** (Graphics and Machine) mode. **TurnMaster** would receive a "***Run Time Error 5"*** and shut down.

40. Disable the ***CNC Edit Window*** while a program is running. This prevents the user from changing the **.CNC** program during execution.

41. If the user cancels the "**Program Pause**" screen, it will now behave as if the "**Stop Program**" button was pressed.

Date: November 26, 2008
Subject: Enhancements resulting in revision 4.3.74
By: Jeff Kidd

1. Corrected a problem with ***"Get Settings"*** button on the **Machine Parameters** page, **Serial Spindle** tab. Occasionally, the wrong serial port was being tested.

2. Improved the reliability of the Serial Spindle "***Get Settings***" button in **Machine Parameters**.

3. Improved the Global Error Message to be more descriptive of the error and calling routine.

4. Improved both the G-Code Parser's and Math Evaluator's syntax error messages to include the line # and source text of the G-code line containing the error.

5. Improved the handling of MN400 Communications Errors using the USB port. **TurnMaster** will now try to diagnose the problem and reconnect if possible.

6. Corrected a problem after an **M66** command while in *Relative Programming Mode*. The next computed destination was not always correct.

Date: September 18 2008
Subject: Enhancements resulting in revision 4.3.72
By: Jeff Kidd

1. The threshold for displaying the *"Tool not at start."* message at the beginning of a program was changed from 0.00005 inch to the distance each axis moves per motor step.

2. The **File|Open** and **File|Close** menu selections now query the user to save the current CNC program if it was not saved.

3. **TurnMaster** will no longer automatically save the current CNC program file when switching from the large ***CNC Edit Mode*** to the normal view mode.

4. Modified the Diagnostics to use a safe speed when moving ***away*** from the limit switch. This assures adequate distance for the experimental high-speed move back to the limit switch.

Date: July 17 2008
Subject: Enhancements resulting in revision 4.3.71
By: Jeff Kidd

1. If running **TurnMaster** on a Laptop PC, don't automatically turn ***NumLock*** on when entering the MN400 Jog screen.

   Previously, NumLock was turned on when entering the MN400 Jog screen and restored to it's previous state upon exiting. Now this only happens if the PC is a Desktop PC.

2. For backward compatibility the * (multiply) operator is now supported with or without being surrounded by square brackets.

3. Under the ***Tools | Threading*** function, **TurnMaster** now uses the **Unified Standard (UN)** depth ratio for internal threading and the **Unified Standard - Rounded Crests (UNR)** depth ratio for external threading, respectively, from the ***Machinery's Handbook 27****, p. 1734*. Previously, the **Sharp V-Thread** ratio was being used for both internal and external threading.

4. Changed the Measured Move values in the MN400 Jog screen to be user definable.

Date: May 19, 2008
Subject: Enhancements resulting in revision 4.3.69
By: Jeff Kidd

1. Corrected a problem in the large editor window, if **File|Save** was selected with an unnamed file, TurnMaster would not ask for a "Save As" filename.

2. Corrected a problem in the **File|Save As** window. If <Esc> was pressed an error occurred.

3. Improved the drawing of the last move in **Graphics** mode. Occasionally, when starting a new program, TurnMaster would redraw the last move from the previous program execution.

4. TurnMaster no longer displays the "End of Program" message if the part program was run in graphics mode only.

5. Corrected a problem in the **G25** command. If an L-parameter (# of Loops) was specified using a variable, then the subroutine would only execute once. For example if **(INCVAR)** was set to the value 5, then the command **G25 #SUB1 L(INCVAR)** would call **SUB1** 5 times.

6. Corrected a problem in the **G01** command. If the move was at a very slow speed, **TurnMaster** would sometimes hang up waiting for the move to complete.

7. Modified the **Tool Chest** form to display tool specifications in a table format.

8. Corrected a problem on the MN400 Jog Screen, where pressing the X or Z keys did not always update the respective counter to 0 on the first attempt.

9. Updated the "Speeds & Feeds" process under the Tools menu to better support the wide variety of tools and materials.

10. Added a Status message indicating the Pass # in progress for Canned Cycle commands (eg. G33, G81, and G82).

11. Corrected a problem when the Z-Axis labels were not always visible after a screen pan.

Date: April 7, 2008
Subject: Enhancements resulting in revision 4.3.65
By: Jeff Kidd

1. Corrected a problem in the Diagnostics. If the initial unramped move failed on a particular axis, the max unramped and max ramped speeds for that axis would sometimes be incorrectly reported.

2. Corrected a problem in the parser if a <Tab> character was seen in the G-code program, the parser incorrectly reported an error.

3. Corrected a problem where if the <Esc> key was pressed while in the "File Open" dialog box, an error would occur.

4. The G04 (Dwell) command uses the Status Bar to count down the seconds remaining, but does not restore the Status Bar after completing. This has been corrected.

5. On the KeyPause form, the <Step> button is now the default. Previously, the <Continue> button was the default.

6. The full Revision # is now displayed at the top of the main form. For example "TurnMaster Pro 4.3.65" instead of "TurnMaster Pro 4.3".

7. Corrected a problem when in Step mode, after a program completed, the top bar menu items sometimes remained disabled.

Date: March 04, 2008
Subject: Enhancements resulting in revision 4.3.64
By: Jeff Kidd

1. Allow variables and formulas in the "**S**" (Spindle Speed) parameter of the M03 and M04 commands. For example:

```
        G28 (NormalSpeed) = 100     /-- Set normal Spindle Speed
        M03 S(NormalSpeed)          /-- Start CW.  Speed = 100
RPM
        M04 S[(NormalSpeed)*2]      /-- Start CCW.  Speed = 200
RPM
```

2. Corrected a problem with M06 (Tool Change), and G04 (Dwell) when used with Continuous Contouring. Previously, the M06 or G04 did not clear the pending machine moves before executing the Tool Change or Dwell.

3. Added an informational message when converting a DXF file if there are any unsupported drawing entities present. Only LINE, POLYLINE, ARC, and CIRCLE entities are supported. All other more complex entities should be converted to lines and arcs before exporting to a DXF file.

   Note: In ***CorelDRAW***ä use the **Arrange | Ungroup All** function to separate any lines and arcs into individual entities. Also use the **Arrange | Convert to Curves** function to convert any text to lines and arcs.

4. Corrected a problem related to the 7300CV VFD. If the motor rated frequency was not 60 Hz, the wrong speed (RPM) was displayed on the "Outputs:" caption of the Main Menu.

5. Increased the number of Comm ports number available for the MN400 from 4 to 8 to allow the use of USB to Serial converter cables.

6. Corrected an "***Overflow: Error 6***" problem when testing Input #8 using the "***M97 i8***" command.

7. Improved the Diagnostics process:

    1. added ***Show Last*** results button.
    2. added separate distances for Ramped vs Unramped tests.
    3. improved the homing speed during the tests.

8. Corrected several problems on Vista and XP environments using the
   MN400 with USB connection, **TurnMaster** would:

   - occasionally misinterpret the current X position.
   - occasionally start a subsequent move before the current move was completed.
   - receive an Overflow Error when executing **G02**, **G03** commands.

   These problems affected Versions 4.3.38 to 4.3.53.

9. Added toggle buttons on the main form for Ouput #7 (Aux 3), and Output #8 (Aux 4).

10. Added the following M codes to work with Outputs #7 and #8 (Aux 3, and Aux 4):

    M41 (Aux 3 On)
    M42 (Aux 3 Off)
    M43 (Aux 4 On)
    M44 (Aux 4 Off)

Date: December 17, 2007
Subject: Enhancements resulting in revision 4.3.53
By: Jeff Kidd

1. Allow the selection of Baud Rate and Communication Settings for both the Fluxmaster 100 and the Teco 7300CV spindle controllers.

2. Added a button on the “Machine Parameters” form to automatically detect communication settings for serial spindle controllers.

3. TurnMaster no longer displays communications error messages if the software is not unlocked.

4. Num Lock is automatically turned on when entering the MN400 Jog form in order to allow the arrow keys on the keypad to act as jog keys. Num Lock is restored to its previous state upon exiting the MN400 Jog form.

5. Added new command **M29 Home to Limits**. The format is:

   M29 *axis* (#varname)

   where ***axis*** can be any axis (x or z). If in absolute programming mode then the x or z position is returned in **#varname**. Otherwise the incremental movement (the distance from

the last position to the limit) is returned in **#varname**.

6. Corrected a problem where the MN400 would sometimes initialize at the wrong startup position at the beginning of program start.

7. After pressing the Pan icon, the Mousepointer changes to a hand. Additionally, the "zooming rectangle" is no longer drawn when in Pan mode.

8. Corrected a problem for the OptiStep controller on the **Move to Point** screen. If ***<Alt> A*** was used to press the Accept button, focus would not return back to the screen.

9. Corrected a problem with the FM100 spindle controller. Occasionally an erroneous timeout message would display waiting for the spindle speed to arrive at the desired RPM.

Date: November 06, 2007
Subject: Enhancements resulting in revision 4.3.41
By: Jeff Kidd

1. Enhanced the warning message when running a program with a serial control spindle.

2. Previously, the "Tested" flags were not being cleared when a change was made to the CNC program. This had the effect of not displaying an appropriate warning message. This has been corrected.

3. Anti-Virus message will no longer display if Operating System is later than Windows 98 and therefore support for OptiStep controller not needed.

4. Added logical operators **=**, **>**, **<**, **>=**, and **<=** to the math evaluator. If a formula is true, the math evaluator returns **-1**, if the formula is false, **0** is returned.

5. The Fluxmaster Spindle controller will now display the correct Spindle RPM on the front panel based on the gear selected. Previously it displayed the motor RPM.

6. Corrected an "Overflow" error during ramped moves if IN7 or IN8 signals were active.

7. Added support for the Teco 7300CV Spindle Controller.

8. Increased number of external outputs from 6 to 8 on the outputs tab in machine parameters for the MN400 Controller.

9. Added the ability to lock and unlock the tool changer using external outputs 2-8. (previously this worked on output #3)

10. Previously under certain conditions, the Main Menu was not being
re-enabled after a program was run. This has been corrected.

Date: Sept 4, 2007
Subject: Enhancements resulting in revision 4.3.24
By: Jeff Kidd

1. Check for valid I, K parameters which represent the Center Point
   Offsets during circular moves.

2. Clear Continuous Contouring buffer when M00 Pause command seen.

3. Corrected a problem with the “Program Paused” screen which would sporatically disappear.

4. Corrected a problem with the <Stop> and <Pause> buttons on the toolbar being ignored while a program was running.

5. Added TMPFW.CHM Help File for Vista PC users.

Date: June 7, 2007
Subject: Enhancements resulting in revision 4.3.16
By: Jeff Kidd

1. Corrected a problem in M06 (Change Tool). If no tool was selected in the command line, and <Cancel> was pressed on the Tool Selection Input Box, an error occurred.

2. Allow for very slow feed rates on MN400 jog. Limit the minimum feed rate to 0.2 IPM on jog screen. Previously, if a very slow feed rate was entered, the Position Counter may have displayed an incorrect value.

3. Corrected a problem with arithmetic divide. If the denominator was a user-defined variable with a negative value, the result would always be 0.

4. Improved the reliability of the Diagnostics by slowing down the “Home to Limit” movement so that an overspeed condition does not occur.

5. Modify the **G82 Canned Cycle For Reducing Part Length** function as follows:
   - Allow **I** parameter (Z-Distance per cut). If **I** is specified, the number of cuts (**Q** Parameter) is computed.
   - Allow **X** parameter of 0.
   - Allow **Z** parameter to be either positive (cut to the right) or negative (cut to the left).

6. Display the default tool number and description in the status
   bar on startup.

7. Allow user-defined variables and arithmetic functions in the
   parameters of M-Codes. Previously only G-Codes supported these functions.

8. Added Pan and Zoom toolbar buttons. If the Pan button is <Pressed>, then the Zoom button will be <Unpressed> and a Mouse Drag will cause the screen to Pan. If the Zoom button is <Pressed>, then the Pan button will be <Unpressed> and a Mouse Drag event will cause the screen to Zoom.

9. Improved the wait times while Loading and Saving very large CNC files.

Date: Apr 2, 2007
Subject: Enhancements resulting in revision 4.3.8
By: Jeff Kidd

1. Added Constant Surface Speed (CSS) functionality for the MN400 controller using the Fluxmaster 100 spindle controller. **G96 Sxxx** will set a constant surface speed of **xxx** ft/min. **G97 Sxxx** will cancel CSS and set a constant RPM of **xxx**. **G97** without an S-Param will cancel CSS and retain the current RPM setting.

Date: Mar 29, 2007
Subject: Enhancements resulting in revision 4.3.7
By: Jeff Kidd

1. Corrected problem with Backlash reinitializing at inappropriate times.

2. End current “Continuous Contouring” stream when M00 (Temporary Stop) or M06 (Tool Change) encountered.

3. Corrected Error message when Fluxmaster spooling up.

4. Reduced number of retries if Fluxmaster is not available.

5. Corrected bug detecting Windows VISTA Operating System.

6. Corrected bug detecting USB connection to MN400 Controller.

7. Corrected Overflow error if Backlash value was too large.

8. Corrected problem updating the counters if Backlash used.

9. Retain “Measured Move” values after exiting MN400 Jog screen.

10. Corrected bug when TMPFW does not prompt to save an unsaved file.

11. Corrected bug when “Save As” not marking file as saved.

12. Clean up screen graphics on Machine Parameters – Jog Tab.

13. Corrected “Run time error 5” if multiple pauses and restarts were issued during a machine move.

14. If the Spindle button pressed while in Graphics mode, we now give the user the option of turning on Machine mode. Previously, pressing the button would do nothing in Graphics mode.

15. If File | Open selected, then cancelled, do not ask the user to “Restore Graphics ?”.

16. Do not reset the counters after “Restore Graphics”.

17. Corrected a problem during a Threading operation with the Fluxmaster. Previously, there was up to a one minute delay between threading passes.

18. Retain the last directory used in a “File | Open” menu selection.