

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 via FAX at (770) 422-7854 or on the WEB at <http://www.microkinetics.com/>

\*\*\*\*\*

#### MillMaster Pro 2014 for Windows

Date: March 24, 2014 Subject: Enhancements resulting in revision 6.0.006 By: Jeff Kidd
--

1. Improved the speed of drawing the graphics of large CNC programs with numerous small circles and lines.
2. Corrected a problem on the **Diagnostics Results** screen. If all 3 axes were tested, then give the user the option to save the results into the **Machine Parameters**.
3. Corrected a problem in the **Pocketing Rectangles and Circles** screen. Certain input parameters were causing the screen to hang up.
4. Corrected the erroneous display of "*Invalid Tool Number ...*" during an **M06** command if the selected tool did not have a textual description.
5. In the **Wizards | Find Part Edges** screen, allow user to "log" the XY coordinates of found center points to the end of the current CNC program.
6. Modified the CNC display to highlight the 4th line (instead of the 3rd line) if the text size is small.
7. Enhanced the **Wizards | Collect Part Geometry** to write out .DXF files compatible with **BobCAD v25**.
8. Update the **Wizards | Collect Part Geometry** and **Wizards | Find Part Edges** screens to display distances in *MM* instead of *Inches* when in Metric mode.
9. 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.
10. Improved the wording on the **Teco 7300** VFD communication error message box to provide clearer and more useful information.
11. Improved the reliability of the **Get Settings** button on the **Machine Parameters** screen, **Serial Spindle** tab.

12. 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.CV7** will be sent to the **Teco 7300** VFD.  
**MK Mill.CV7** will be included in the installation package.
13. 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.
14. Corrected a bug in the **Tool Chest** screen, *Associate Image File* button. Sometimes the wrong initial search directory would appear. Now, the search directory will always default to the application directory.
15. Corrected a problem with **Tool Name** descriptions, which were being converted to all lower-case.
16. Corrected a problem when a **G25** (Call Subroutine) command. If an invalid subroutine name was entered, the CNC program would be erroneously truncated.
17. 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.
18. Added "L" (log) hotkey on the MN400 Jog Screen. When "L" is pressed, the current (X, Y) position will be logged at the end of the current program to aid in programming.
19. 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.
20. Added the read-only variable **MM\_ToolNum**, which returns the currently active tool number (1-99).
21. Added checkbox "*Prompt for Return to Start?*" on the **Machine Parameters** screen. The default value is checked. If this checkbox is unchecked then MillMaster will no longer ask the user "*Return Tool to Start ?*" when running a program.
22. Corrected a problem with the graphical tool during a **G00** (Rapid Move) command. If the **Z**-value was  $< 0$ , the graphical tool image would not always display correctly.
23. Improved the drawing of the Z-axis ruler. Previously, when using very thin material, the ruler would not show any ruler lines.

Date: May 29, 2013

Subject: Enhancements resulting in revision 5.2.355

By: Jeff Kidd

1. Corrected a problem with **G79**, **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. The **M06 - Tool Change** message box is now larger and easier to read.
3. Improved the drawing of Z-depths. If a Z-axis move is being made by lifting the tool, don't redraw a lighter shade of green for this area.
4. Added a **Single-Step** checkbox to the **M06 - Tool Change** message box. If checked, **MillMaster** will enter Single-Step mode and stop before the execution of each line.
5. Added read-only variables (**MM\_TOOLDIAMETER**), (**MM\_TOOLLENGTH**), (**MM\_TOOLXOFFSET**), and (**MM\_TOOLYOFFSET**) which will return the diameter, length, and offsets of the currently selected tool, respectively. See *Appendix E* in the user's manual for more information.
6. Improved the display screen of syntax errors related to math operations found in the G-code program.
7. Corrected the problem where the main display redraws after switching to the full-screen editor when any change to the program was made.
8. Added <Cancel> button on the "Save System Parameters?" screen. If pressed, no action will be taken, and **MillMaster** will not terminate.
9. Added **Start Z-Level**, **Plunge Feed Rate** and **Cut Feed Rate** input fields on the Pocketing Wizard screen. Added a G0 Z.05 to go up and out of the cavity at the end.
10. Previously, When loading a new part program but not redrawing the screen, if the new program changes the diameter of the current tool with meta commands, the tool shape is not shown. This has been corrected.
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 5.2.339

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 displays for total running time, and if a program terminated abnormally.
3. If an **R** (Radius) parameter is listed in a **G02/G03** command, ignore any leftover **I**, **J**, or **K** parameters from previous cycle commands. In addition, **G80** (Cancel Cycle) now clears all **I**, **J**, **K**, and **Q** values from previous cycle commands.
4. The "x origin" value in the setup file (\*.tmp) is now being saved as "Left", "Right" or "Center" instead of 1, -1, or 2. The "y origin" value now being saved as "Front", "Back" or "Center" instead of 1, -1, or 2. The "z origin" value is now being saved as "Top" or "Bottom" instead of 1, or -1. Setup files using the old format continue to be supported.
5. If a program is cancelled via the **Key Pause** screen, then **MillMaster** 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  $(10 * \text{Inches})/\text{Min}$ , regardless of **Inches/Metric** mode. No changes are made to the inch mode.
7. Corrected a problem with the *"Estimated Running Time"* for some multi-depth canned cycles in *Graphics* mode. The estimated times for these commands were too high.

Date: September 13, 2012

Subject: Enhancements resulting in revision 5.2.333

By: Jeff Kidd

1. If **MillMaster** was loaded by double-clicking on a **.CNC** file in Windows Explorer, that file will now be added to the MRU list.
2. If a newly loaded program changed any screen settings via *MetaCommands*, then the "Restore Screen ?" message box will notify the user that "Screen settings have changed."
3. Display the current **.CNC** filename when printing the screen image from the **File | Print** menu.
4. Corrected a problem when the **G77/G78** commands. If the **Q**-parameter and respective **I** and **J** parameters were missing, an error message was sometimes not being displayed.
5. During **G02/G03** commands, only check **I**, **J**, and **K** arc errors if the **Run | Check Syntax** option is checked.
6. Added a "No pause on comment lines when stepping." checkbox on the *KeyPause* screen. The default value for this checkbox is *checked*.
7. If nothing changed on the **Options | Material Size** screen, then don't redraw main screen.
8. 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.
9. 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.
10. Corrected a problem with the **G04 Dwell** command, which was not allowing a user keystroke to pause the program.

Date: July 10, 2012

Subject: Enhancements resulting in revision 5.2.327

By: Jeff Kidd

1. Do not stop the **Spindle** or **Coolant** during tool changes if the *Tool Change Pause* checkbox is not checked.
2. Corrected a problem drawing the main viewport. Sometimes, not all of the Y-axis labels would display.
3. Modified the **G77**, **G78**, and **G79** *Deep Channel Canned Cycle* commands. The active axis letter (X for G77, Y for G78, and Z for G79) specifies the cut depth and is not an actual coordinate. It is assumed that the tool is 0.05" from the surface to be cut, on the respective axis, before the command is issued. The other axes (Y and Z for G77, X and Z for G78, and X and Y for G79) are actual coordinates in *Absolute* or *Incremental* based on the current mode.

Additionally, the **G77** now uses the **I**-parameter to represent **X**-axis increment. **G78** uses the **J**-parameter for **Y**-axis increment. **G79** continues to use the **K**-parameter for **Z**-axis increment. Previously, all three commands used the **K**-parameter.

Date: May 10, 2012

Subject: Enhancements resulting in revision 5.2.323

By: Jeff Kidd

1. In the **Machine Parameters** screen, added the ability *Save As* a setup file in a different directory. Previously, *Save As* only worked on the current working directory.
2. Allow "Single-Step with Canned Cycles?" checkbox on the *Program Pause* screen to work in Graphics Only mode.
3. Corrected a problem with the **{MATERIAL SHAPE}** metacommand. Previously, if "RECTANGLE" was specified, and the current shape was "ROUND", then the desired shape did not get set.
4. Changed the verbiage on the **Options | Material Size** screen from "Circle" to "Round" to match the corresponding metacommand.
5. Added a checkbox "*Write values to part program?*" to the **Options | Tool Start Positions** screen. If checked, the X, Y, Z, and A start positions will be written, as metacommands, to the part program.
6. Added D-parameter for **G77** and **G78** cycle codes. If present, the cutting speed for the **X** (for G77) or **Y** (for G78) axes, respectively will be at the commanded rate in inches per minute.
7. Corrected a problem using the Q-parameter with **G77**, **G78**, or **G79** cycle codes. Under certain conditions, the incremental depth was not being calculated correctly.
8. Corrected a problem when using "Absolute Z-Position" for tool changes. If the Z-position was already near the limit switch and a tool changed to a longer tool was completed, MillMaster would try to restore the original Z-position with the longer tool, which could result in hitting the limit switch. This problem has been corrected.
9. Added *read-only* access to Tool Diameters, Lengths and Offsets. Use the system variable names **MM\_TOOLDIAMETER<sub>x</sub>**, **MM\_TOOLLENGTH<sub>x</sub>**, **MM\_TOOLXOFFSET<sub>x</sub>**, and **MM\_TOOLYOFFSET<sub>x</sub>**, where x is a valid tool number from 1 to 99. See **Appendix E** in the User's Manual for a complete list of system variables.

10. 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"**

11. Added ability to modify the order of axis movement during an M25 command. Normally the order is Z, Y, X if the Z-position is lower than the start position, or Y, X, Z otherwise. You can now specify the order after the M25 command. If any axes are left, they will be done in the default order.

For example:       M25 XYZ       /-- Move X, Y, then Z-axes.  
                  M25 Z         /-- Always move Z-axis first.  
                  M25         /-- Use default order of axes.

12. 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 23, 2012

Subject: Enhancements resulting in revision 5.2.308

By: Jeff Kidd

1. Added ability to "Collect Geometry" for an *Island* or *Boss*, via the **Wizards | Collect Geometry** screen.
2. Added G68 and G69 commands. G68 will "Collect Geometry" for a *Pocket*. G69 will collect geometry for an *Island* or *Boss*. See the documentation for a detailed explanation.
3. Added "Z to Tool Change Pos" button to the **MN400 Jog Screen**. This button will only appear if the "Use Absolute Z-Pos" checkbox is checked on the *Advanced* tab of the **Machine Parameters** screen. If pressed the Z-axis will move to the predetermined tool change position, regardless of which tool is active.
4. Added <Alt> Z keystroke on the **MN400 Jog** screen. When pressed, the current Z-position will be stored as *Z-Absolute Tool Position*, as entered on the **Machine Parameters** screen, *Advanced* tab.
5. Corrected a problem on the **Machine Parameters** screen. If *Feed Hold* mode was changed, the value was being saved, but the MN400 was not being reconfigured until **MillMaster** was restarted. Now, the MN400 is reconfigured with the correct values immediately.
6. When *Feed Hold* mode is turned off, and the safety shield is opened, stop rotation of the VFD in addition to stopping X, Y, and Z movement. The VFD is restored if the <Continue> button is pressed on the *Pause* screen.
7. Corrected a problem when a G-Code program is running past midnight. Under certain rare conditions, **MillMaster** would not handle the date change correctly and hang.
8. Added an optional "*Extra Delay During Machine Moves*" value in the **Advanced** tab of the **Machine Parameters** screen. A very fast PC executing moves for a long period can read the counters too often and possibly overstep some data causing the program to hang. A value of .003 to 0.005 (in seconds) appears to solve this issue.
9. Added read-only access to most system variables in the **Machine Parameters** screen. See the new **Appendix E** in the User's Manual for a complete list of variables.
10. Improved the graphics on the **File | Import DXF** screen. Previously, the display of the imported DXF file was not always centered or scaled properly.

Date: November 30, 2011

Subject: Enhancements resulting in revision 5.2.269

By: Jeff Kidd

1. Corrected a problem after all cycle codes (G77-G89). Sometimes the Graphical Tool Z-position was not being displayed correctly.
2. Added *Use Absolute Z-Position* checkbox in the Options | Machine Parameters screen, *Advanced* tab, *Tool Change* frame. If checked, all tool changes will go to the specified Z-position. This value should be the Z-position desired calibrated with Tool #1.
3. Corrected a problem when selecting Baud Rate for MN400 using serial communications. Selected speeds greater than 9600 baud were being ignored. Note, this was not a problem when using the MN400 with USB communications.
4. Corrected 2 problems with the MN400 when connected using serial communications. When in the *MN400 Jog Screen*, and a fixed length move was started, pressing a keystroke was not pausing the motors properly. In the *Move to Point* screen, after pausing the motors once and restarting, subsequent keystrokes would not pause the motors. Note, this was not a problem when using the MN400 with USB communications.
5. 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 MillMaster would shut down. That error now is handled with an error message.
6. 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.
7. Corrected a problem when pausing tool motion via the keyboard when using an MN400 with a serial connection. In the **MN400 Jog** and the **Move To Point** screens, a keypress was sometimes being ignored.
8. Improved the tool side view flicker when executing cycle codes in Graphics mode.

Date: October 14, 2011  
Subject: Enhancements resulting in revision 5.2.263  
By: Jeff Kidd

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

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

2. 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.
3. Added **{defvar}**: MetaCommand. This is a load time command that is similar in functionality to the **G28** runtime command. When present, the specified variable will be set when the program loads. The variable will have this initial value regardless of what line the program is started from.

For example to set the "depth" variable to -0.5 and the Feed Rate Multiplier "FRM" to 4, use the following:

```
/* {defvar}: depth,-0.5  
/* {DEFVAR}: FRM,5
```

4. The main view now can display negative **X** and **Y** grid values as necessary. Previously, only positive grid values were displayed.
5. Added **D**-parameter (Downfeed Speed) to the **G79 Deep Channel Canned Cycle** command. If present, movement on the **Z**-axis into the material will be at the rate specified in the **D**-parameter. The format is the same as the **F**-parameter. Movements along **X** and **Y** axes are unaffected by the **D**-parameter.

For example:

```
G79 x1 y1 q4 f90 d8 /-- Canned Cycle  
/-- X and Y feedrate is 9.0 inches per min.  
/-- Z federate into uncut material is 0.8 inches per min.
```

Date: August 31, 2011

Subject: Enhancements resulting in revision 5.2.258

By: Jeff Kidd

1. Corrected a problem when commanding an **A**-axis move with a **V** (velocity) parameter. In some cases the V-parameter was being ignored.
2. Added *A-Axis Speed* (RPMs) entry box for Rotary Turrets on the MN400 Jog Screen. The A-Axis will move at the RPM speed entered if the **Feed** radio button is selected.
3. 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.
4. Improved the error handling on the **Teco 7300** spindle controller. If a speed greater than the maximum allowable speed is issued, **MillMaster** was waiting unnecessarily before executing the next G-Code command.
5. When an **M00** command is processed, display the comment (if any) in the message box. This allows the user to customize the message.
6. If the "Write Values to Part Program" checkbox is checked in **Options | Material Size** menu, write out the *{UNITS}*: metacode prior to writing the part dimensions. This corrects a problem if the metacode sets the part dimensions in one unit of measure, and the program startup is assuming the other unit.
7. Corrected a problem when switching from *Inches* to *Millimeters* mode. The Z-grid was not being redrawn correctly.
8. Disabled line autowrap from the editor Display. Previously, a line wrapped, the displayed line # would not match the actual line # when the program executed.
9. Added a message box, reporting estimated machining time, at the end of a program run in *Graphics* mode.
10. 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.
11. 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.
12. Added **G77** and **G78** commands. These are similar to **G79**, only the primary axis being acted on are **X** (for **G77**) and **Y** (for **G78**) instead of the **Z**-axis used with **G79**.

13. Corrected a problem when changing tools. If a **G92** (Set current position) is in effect and an **M06** (Tool Change) was seen with the new tool having different **X** and/or **Y** values, the initial jog of the tool position was not taking into account the **G92** values.
14. 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 5.2.238

By: Jeff Kidd

1. In *Graphics* or *Both* mode, allow pause via keyboard entry when drawing an arc or circle.
2. 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.
3. 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.
4. 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.
5. **Absolute Programming Mode** (G90) is now the default for **MillMaster** if no G90 or G91 command is present. 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. **Multiple Arc Programming Mode** (G75) is now the default mode for **MillMaster**. Previously **Single Arc Programming Mode** (G74) was the default mode.
7. Corrected a problem when the screen was unnecessarily being redrawn when running a CNC program that contained Meta Commands.
8. For consistency between programs, **Edit | Undo** is now the first selection listed under the **Edit** menu.
9. Added support for *Incremental Programming Mode* on the **Wizards | Pocketing** screen.
10. The Default Feed Rate on the **MN400 Jog Screen** is changed from 10 ipm to 5 ipm.
11. Added *Calibrate Probe* functionality to the **Wizards | Find Part Edges** screen.
12. Added **Wizards | Collect Part Geometry** screen. This wizard will collect points around a circular, rectangular, or other smooth shaped cutout (pocket).

13. Improved the **File | DXF Import** function as follows:

- Coordinates are written with up to 5 decimal places.
- G92 is written as a comment. If the G92 statement is needed, the user can uncomment it.
- Extra comments are written to the output file.
- Points entities are now correctly interpreted in .DXF files.

Date: March 22, 2011

Subject: Enhancements resulting in revision 5.2.222

By: Jeff Kidd

1. Corrected a problem with the **G79 Deep Channel Canned Cycle**. If the X or Y values were not an even multiple of distance per step for the respective axes, the channel would not be cut correctly.
2. Retain screen values from the **Wizards | Pocketing** screen and the **Wizards | Bolt Hole Circle** screen.
3. Removed "(inches)" from the caption for "Tool Change Extra Clearance" on the **Machine Parameters** screen.
4. The Z-Axis side view of the tool, to the right of the main display, now shows the relative thickness of the cuts in the material.
5. 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.
6. 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.
7. Improved the drawing of the grid. When the number of units was very large, too many grid lines were being drawn.
8. 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.
9. Corrected a problem when opening a .CNC file from the Windows File Folder, **MillMaster Pro** would not always auto-load the file selected.
10. Corrected a problem when a new tool was added to the tool chest, the graphics display would sometimes draw incorrectly.
11. Corrected a problem when the Serial Spindle was sometimes losing communication after visiting the **Machine Parameters** screen.



Date: December 29, 2010 Subject: Enhancements resulting in revision 5.1.205 By: Jeff Kidd
---

1. Corrected a problem with the **M06** (Tool Change) command. If the new tool had different **X** and/or **Y** offsets as the old tool but the lengths of the tools were the same, then the **X, Y** offsets would not take effect.
2. 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 m1000,500,500**  
                      **m94 "m1000,500,500"**  
                      **m94 m1000,500,500 /MN400 native move**

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

3. Added support for the **R** (Radius) Parameter in the **G02/G03** when in **G18** (XZ Coordinate) and **G19** (YZ Coordinate) mode. If the **I, J** and **K** parameters are not specified and the **R** parameter is specified, **MillMaster** 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.
4. Corrected bug in **Full Screen Counter** mode, where the **Loop Counter** and **Full Screen Edit Mode** button were inadvertently displayed.
5. Save the **View | Grid** settings in the appropriate .MPS file.
6. Added # of Chords per Circle to the Advanced tab of the Machine Parameters screen. Previously, **MillMaster** always used 360 chords per circle when using chords.
7. Do not report an error when closing **MillMaster** if the MN400 controller was turned off before exiting.
8. After running diagnostics, the *Diagnostic Results* screen defaults to **No**, when asking whether to save the results.
9. On the **Diagnostics Results** screen, display N/A in the results summary for any axes not tested.
10. Disable entry fields for *Unramped Distance* and *Ramped Distance* for any axes not selected.

11. 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.
12. The **G79**, **G83** and **G87** multisweep cycles now assume that the current Z-position is at 0.05" distance above the start of the drill. This allows cycles to begin below (or above) the surface of the material in case previous operations have already removed material. Previously, all calculations were based on 0.05" above the surface of the uncut material.
13. Corrected a problem with the **K**-parameter in **G79**, **G83** and **G87** multisweep cycles. If the **K**-parameter (depth per pass) was not an even multiple of the total depth, then **MillMaster** would divide the total distance by the computed # of passes instead of using the depth per pass as specified by the user.
14. Corrected a problem with the **{SHEIGHT}**: meta-command. If the value of **{SHEIGHT}**: is different than the *Height* specified in the **Material Size** screen, then the tool Z-position would be displayed incorrectly in the Z-Axis graphics screen just to the right of the main screen.
15. 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.
16. 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.
17. Corrected a problem with multi-axis moves. If one axis was moving much slower than the others, **MillMaster** would sometimes start the next move too soon, which would result in a temporary loss of steps.
18. Don't redraw the **CNC Program Window** after saving the file. This was resulting in the loss of the cursor position within the program.
19. The Z-Axis display to the right of the main screen now shows extra white space below the material. Additionally, a green path is displayed behind the tool showing the maximum depth tool depth seen for this CNC program.

Date: April 20, 2010

Subject: Enhancements resulting in revision 5.1.179

By: Jeff Kidd

1. Speed up the **G79**, **G83** and **G87** (deep hole/channel canned cycles) such that after backing out of intermediate depths, the tool will move back to the last depth at high speed.
2. Improve the **MN400 Jog Screen** to reestablish the correct jog speed after reconnecting to the controller.
3. Corrected a problem in the **M6 - Tool Change** command. If the new tool was longer than the current tool, and the user selected "No" to the "**Continue...**" message box, the original tool along with its positioning values were not restored.
4. Modified the **Return Tool to Start** function as follows: If the current Z-axis value is greater than the Z-axis start position value, then move the X and Y axes to their start positions before moving the Z-Axis. This allows the maximum tool clearance when moving back to the start position.
5. 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.
6. Added the verbage "**Continue ?**" to the **M06 - Tool Change** message box.
7. Added **Write Values to Part Program** checkbox to the **Options | Material Size** menu selection. If checked, meta commands will be written to the header of the .CNC part program to specify the current part dimensions and orientation.
8. Modified the **G79 Deep Channel Canned Cycle** command to cut bi-directionally.
9. Display the currently active tool instead of Tool #1 when initially displaying the **Tool Chest** screen.
10. Do not redraw the screen when exiting the **Tool Chest** screen.
11. Display a message reminding the user to restart **MillMaster** if the backlash values have changed in the **Machine Parameters** screen.
12. Clear any old filenames when performing **File | New**. Previously the last open filename was retained if a subsequent **File | Open** was performed.
13. Corrected a problem when the **Run From 1<sup>st</sup> 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.

14. Added a **Always Run From First Line** button on the **Run From Line #** screen. When pressed the **Run From 1<sup>st</sup> Line** menu item will be checked and all subsequent runs will be from the first line.
15. 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.
16. Corrected a problem with **G02/G03 Circular Moves**. If G02/G03 was the first statement in a program that was not starting from the default start position, the third axis would inadvertently move back to the default start position.
17. 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.
18. Allow user option to skip retrying of the connection to the *Teco 7300 Serial Spindle Controller* for the current session.
19. Improved the accuracy of ARC endpoint calculation on the **File | Import .DXF** function. This corrects a problem where some of the generated **G02/G03** commands could cause a warning message. This warning message states that the center point or endpoints may not be correct.
20. Added up to 9 most recently used (MRU) files on the **File** menu.
21. 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.
22. When opening an existing file, the Line # on the status bar is now set to 1 instead of 0.
23. Corrected a problem in the **Move To Point** screen. If the **<spacebar>** was pressed while the motors were still in motion, a **Runtime Error 5** would sometimes appear.
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.
27. When the **Run From 1<sup>st</sup> Line** menu item is not checked and a VFD spindle controller is installed, if the user selects <Cancel> when asked to enter the line #, the program would continue and present the Automatic Spindle warning message. This problem has been corrected to abort immediately.
28. Added a **Always Run From First Line** button on the **Run From Line #** screen. When pressed the **Run From 1<sup>st</sup> Line** menu item will be checked and all subsequent runs will be from the first line.
29. 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.

Date: October 14, 2009

Subject: Enhancements resulting in revision 5.1.160

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 5.1.154

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. When in *Continuous Contouring* mode, any G-code command with a Z or A-axis movement and no X or Y-axis movement will close the current continuous contouring sequence. Previously, some Z-axis moves were being included in the continuous contouring sequence.
4. Corrected a problem with the G02/G03 *r* (radius) parameter. If the endpoint of the circle was exactly double the radius (resulting in a 180° arc), the parser would sometimes report that "*The R (Radius) parameter specified is too small.*"
5. Added an option on the **Options | Material Size** screen for a round stock material.
6. Added **Wizards | Find Part Edges** which interfaces with a digitizing probe to find part edges and dimensions.
7. Modified the way the *M06 Tool Change* command functions while in *Machine Mode*. If the new tool is longer than the current tool, then the current tool is raised to accommodate the new tool *before* showing the tool change reminder message.  
  
If the new tool is shorter then the new tool is lowered to accommodate the new *after* showing the tool change reminder message.  
  
In previous versions of **MillMaster Pro**, the tool accommodating move was blended into the next G00, G01, G02, or G03 move.
8. Keep track of accumulated partial step amounts during **Jog**, **Move to Point**, and **Interactive Commands**, for use in any subsequent moves.
9. On the Tool Change message box, details of the next tool are displayed for easy tool identification and operator convenience.
10. Added the ability to import *Tool Setup* and *Material Shape* parameters via *meta commands*, generated by **MultiCAM 2009**.

11. Added *Tool Change Extra Clearance* input field in the **Options | Machine Parameters** screen, **Advanced** tab. If entered, the Z-Axis will be raised this additional amount during tool changes.
12. 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.
13. Corrected a problem in the **MN400 Jog Screen**. If the jog speed was very slow (< 0.5 inches/min), the screen counters would sometimes prematurely stop updating.
14. On the **Diagnostics** screen, display a brief summary of the results at the bottom of the scrollable textbox.
15. Corrected a problem when running a program in *Machine* mode. If the user selects **<Cancel>** to the question "*Tool not at start. Auto return to start?*", MillMaster would sometimes still ask the question "*Untested part program should be tested before machining! Machine anyway?*".
16. Added *Loop* counter to the status display above the CNC code window.
17. Removed **Encoder DRO** selection from the **Control** menu.
18. 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 **G92s** 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. Changed the name of the primary Help file from **MMPFW\_MANUAL\_V4X\_A4.PDF** to **MMPFW\_MANUAL.PDF**.
4. 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.
5. 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 **"Continuous Contouring"**. If the (X, Y) destination of move was not an even multiple of machine steps, MillMaster was not accounting for the partial step correctly, resulting in a small accumulation error over time.
4. Added **"Sel"** (Selected Tool) option buttons to the **Tool Chest** screen. Clicking this button next to a tool, then pressing <Save> or <Accept> will have the result equivalent to executing a **"M06 Tt"** CNC command.
5. Set the initial focus on the **Machine Parameters Password** screen to the password entry field.
6. Corrected a problem with the Current Setup File **"Save As"** button on the **Machine Parameters** screen. Before, **MillMaster** would save the file as the name specified, but would not register the new file name as the current setup file on the main form.
7. Corrected a problem with **View | Restore Screen** and **View | Redraw Screen**. After running a program in Graphics mode, the yellow image of the part would sometimes extend beyond the boundaries specified in **Options | Material Size**.
8. When selecting File | New, **MillMaster** no longer displays the message box *"System Parameters have changed..."*.
9. Corrected a problem where the graphical tool on the "Z-Axis" picture box would occasionally move to the top of the picture for no apparent reason.
10. 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.
11. 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.

12. 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, then the user would receive a "Type Mismatch" error and MillMaster would terminate.
13. Added separate fields for A-Axis moves on the MN400 Jog Screen.
14. Corrected a problem reporting very small A-axis positions. If the final position was < 10 motor steps from 0, MillMaster displayed a final value of 0.\
15. Added **Coolant On/Off**, and **Spindle On/Off** buttons to the **MN400 Jog**, and the **Move to Point** screens.
16. Improved the zooming algorithm. Sometimes after a zoom, the X-Y aspect ratio was not correct.
17. Corrected a problem while loading a G-Code program, or while switching between edit and graphics mode, an unexpected dashed rectangle would sometimes appear on the screen.
18. If **Control | Reprogram Limit Sensors** was executed, MillMaster will remind the user to "Save Parameters?" before exiting.
19. MillMaster no longer redraws the grid after running a program in Graphics mode. Previously, this could obscure tool path lines just drawn.
20. In the **File | Open** dialog box, added **All Files (\*.\*)** as an allowable file type in addition to **CNC Files (\*.CNC)**.
21. 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.
22. If the **Material Size** changed, but the values were not saved, then remind the user that "*System Parameters have changed.*" before exiting MillMaster.
23. 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.
24. Improved the function of **Zoom In** and **Zoom Out** buttons. Zooming now maintains the center point of the current view.

25. 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.
26. 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.
27. 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.
28. 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.
29. Added **"Dashed Line when Tool Up"** checkbox in the **Machine Parameters, Advanced** tab. If checked, MillMaster will draw a thin dashed line when the tool is above the material.
30. Increased the size of the CNC Display window to accommodate more lines of the CNC program to be displayed.
31. 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.
32. 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**.
33. 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.
34. 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, **MillMaster** 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.

35. Corrected a problem introduced in Version 4.3.74 where the **Continuous Countouring** and **Track External Control Panel** options, in the Machine Parameters screen, were being inadvertently cleared out.
36. Added a reminder message to save system parameters after executing **Control / Reprogram from Limit Sensors**.
37. Added options on the **Machine Parameters** screen **Machine** tab, to center the origin on the X and/or Y axis. Previously the origin was either to the left/right, and at the top/bottom of the part.
38. 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 **MillMaster** would be relative to that tool instead of Tool #1. **MillMaster** will now prompt the user to change the active tool and touching off Tool #1 before touching off any other tool.
39. Added a one-time "Terms & Conditions" notice.
40. Display an "Controller Offline" message if user tries to perform a function that requires connection to the controller, and it is Offline.
41. Corrected a problem with **Control / Reposition from Limits, Control / Reprogram from Limit Sensors** and **Control / Home to Limits**. If the user aborted a move in progress, then the CNC Edit Window would revert to the last saved copy the CNC file, losing any changes since the last save.
42. Added a validity check for I, J, K values on **G02/G03** commands.
43. 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.
44. Corrected a problem when importing a **.DXF** file when using a **Torch/Plasma Mill**. The Torch was not consistently turning on during **G02/G03** moves.
45. Disable the **CNC Edit Window** while a program is running. This prevents the user from changing the **.CNC** program during execution.

Date: November 26, 2008

Subject: Enhancements resulting in revision 4.3.83

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. **MillMaster** 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 12, 2008

Subject: Enhancements resulting in revision 4.3.81

By: Jeff Kidd

1. Corrected a problem with **Continuous Contouring**. If the program was paused via the keyboard and "Stop Program" was selected, **MillMaster** would sometimes make one extra move before quitting.
2. Occasionally, due to very small rounding errors, **MillMaster** would display a "Tool not at start." message when, in fact, all axes were at the start. Previously the threshold for displaying the "Tool not at start." message was an arbitrary 0.00001 inch. This threshold was changed to equal the distance each axis moves per motor step, so that the message will only display if the motors are at least one full step from their respective start.
3. **MillMaster** will no longer automatically save the current CNC program file when switching from the large **CNC Edit Mode** to the normal view mode.
4. The **File|Open** and **File|Close** menu selections now query the user to save the current CNC program if it was not saved.
5. In Graphics mode, if a Z-down is commanded, then an immediate Z-Up without an intermediate move, **MillMaster** now displays the hole. Previously, nothing was displayed.
6. Enhanced the HPGL to DXF conversion utility to correctly interpret the following non-standard HPGL commands.
  - PP (Pen Peck)
  - TC (Tool Change)
7. Enhanced the **Tools | Bolt Hole Circle** worksheet to include type of canned cycle, federate, dwell, final depth, and incremental depth when generating G-Code. Also added the ability to generate code in incremental as well as absolute programming mode.
8. 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.
9. Modified the Tool Change message to be compatible with CIRCAD™ generated .CNC files.
10. Added an option in the **Machine Parameters, Advanced Tab** to allow curves to be approximated by multiple chords.

Date: July 17 2008

Subject: Enhancements resulting in revision 4.3.78

By: Jeff Kidd

1. Corrected a problem in the **Engrave Text** function under the **Run** menu. Starting with Version 4.3.55, the text size was being ignored.
2. For backward compatibility the \* (multiply) operator is now supported with or without being surrounded by square brackets.
3. If running **MillMaster** 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.

4. Corrected a problem when exiting **MillMaster** after jogging. When returning to the start position, occasionally all axes would move simultaneously instead of one at a time.
5. Corrected a problem after zeroing out the X, Y, Z, or A axis in the MN400 Jog screen, **MillMaster** sometimes retained the old value of the axis when computing the next move.
6. Changed the Measured Move values in the MN400 Jog screen to be user definable.



Date: May 20, 2008

Subject: Enhancements resulting in revision 4.3.76

By: Jeff Kidd

1. Corrected a problem in the large editor window, if **File|Save** was selected with an unnamed file, **MillMaster** would not ask for a "Save As" filename.
2. 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.
3. Modified the **Tool Chest** form to display tool specifications in a table format.
4. Corrected a problem on the MN400 Jog Screen, where pressing the X, Y, Z, or A keys did not always update the respective counter to 0 on the first attempt.
5. Added a Status message indicating the Pass # in progress for Canned Cycle commands (eg. G79, G83, G85, and G87).
6. Updated the "Speeds & Feeds" process under the Tools menu to better support the wide variety of tools and materials.
7. MillMaster no longer displays the "End of Program" message if the part program was run in graphics mode only.

Date: April 7, 2008

Subject: Enhancements resulting in revision 4.3.71

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 motion was paused and <Cancel> pressed, the motors would sometimes finish the current move before stopping.
4. Corrected a problem if the <Esc> key was pressed while in the "File Open" dialog box, an error would occur.
5. 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.
6. Corrected a problem when trying to Step through an unsaved (and unnamed) G-Code program. Occasionally MillMaster would report "Error 75: Path/file access error".
7. On the KeyPause form, the <Step> button is now the default. Previously, the <Continue> button was the default.
8. The full Revision # is now displayed at the top of the main form. For example "MillMaster Pro 4.3.71" instead of "MillMaster Pro 4.3".
9. Corrected a problem when in Step mode, after a program completed, the top bar menu items sometimes remained disabled.
10. Improved the DXF Import function. Previously, the resulting .CNC file would contain some unnecessary Tool Up/Down movements.

Date: March 04, 2008 Subject: Enhancements resulting in revision 4.3.69 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 an "**Overflow: Error 6**" problem when importing a DXF file containing certain circles.
3. 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.
4. 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** 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.

5. Corrected a problem when a Torch/Plasma machine type selected, the part design was not being drawn even if the torch was on.
6. 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.
7. 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.
8. Corrected an "**Overflow: Error 6**" problem when testing Input #8 using the "**M97 i8**" command.
9. Improved the Diagnostics process:
  - added **Show Last** results button.
  - added separate distances for Ramped vs Unramped tests.
  - improved the homing speed during the tests.

10. Corrected several problems on Vista and XP environments using the MN400 with USB connection, Millmaster would:

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

These problems affected Versions 4.3.33 to 4.3.54.

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

12. 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 4.3.54

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. MillMaster 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, y, or z). If in absolute programming mode then the x, y, 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. Corrected a problem where the diagnostics would occasionally stop before completing all tests.
8. After pressing the Pan icon, the Mousepointer changes to a hand. Additionally, the "zooming rectangle" is no longer drawn when in Pan mode.
9. 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.
10. 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 4.3.42

By: Jeff Kidd

1. Corrected a problem when the Fluxmaster spindle would start automatically when the spindle speed was set with an Sxxx command. Now, if the spindle is stopped an M03 or M04 is required to start the spindle.
2. Enhanced the warning message when running a program with a serial control spindle.
3. 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
4. Anti-Virus message will no longer display if Operating System is later than Windows 98 and therefore support for OptiStep controller not needed.
5. 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.
6. The Fluxmaster Spindle controller will now display the correct RPM on the front panel if the Gear Ratio is changed.
7. Added support for the Teco 7300CV Spindle Controller.
8. Corrected a problem with the automatic tool change.
9. If in "Machine" mode, turn off the external spindle and all outputs when exiting MillMaster.
10. Previously under certain conditions, the Main Menu was not being re-enabled after a program was run. This has been corrected.

Date: August 29, 2007

Subject: Enhancements resulting in revision 4.3.33

By: Jeff Kidd

1. Added ability for the user to Add/Change/Delete User-Defined Materials in the Speeds.MDB database.
2. Added MMPFW.CHM Help File for Vista PC users.
3. Corrected a problem when generating G-Code from a DXF Import. The Z value was not always being set correctly.
4. Correct a problem with MN400 not reinitializing after "Machine Parameters" changed.
5. Corrected a problem with MN400 not detecting MN400 Controller powering On and Off.
6. Clear Continuous Contouring buffer when M00 Pause command seen.
7. Corrected a problem with the "Program Paused" screen which would sporadically disappear.
8. Corrected a problem with the <Stop> and <Pause> buttons on the toolbar being ignored while a program was running.

Date: June 7, 2007  
Subject: Enhancements resulting in revision 4.3.24  
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. Allow user-defined variables and arithmetic functions in the Parameters of M-Codes. Previously only G-Codes supported these functions.
6. 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.
7. Improved the wait times while Loading and Saving very large CNC files.

Date: Apr 9, 2007  
Subject: Enhancements resulting in revision 4.3.9  
By: Jeff Kidd

1. Allow user to select any combination of actions at the start and end of a pattern (either Z-axis move or M8/M9 Torch) during a DXF import.
2. Retain most recently used initial directory for DXF import. Previously



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 "Save As" not marking file as saved.
11. Clean up screen graphics on Machine Parameters - Jog Tab.
12. Corrected "Run time error 5" if multiple pauses and restarts were issued during a machine move.
13. 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.
14. Changed buttons on "Turn Spindle On" message box from Yes/No to OK/Cancel, and changed the default button to <OK>.