

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/>

MultiCAM Mill for Windows

Date: July 02 2008 Subject: Enhancements resulting in revision 1.1.53 By: Jeff Kidd

1. Added **Lathe Style Pocketing** option in the the **Pocketing/Profiling Specs** screen.
2. Added the function **Make CNC Lathe File** from the **CAM** dropdown menu. Documentation is provided in the Help file.
3. Prompt the user to save system parameters before exiting, if any changes were made.
4. Updated the **Drawing Title Block** and the **Tool Selection & Definition** screens to better support a wider variety of tools and materials.
5. Converted the Help file to .PDF for consistency across Windows platforms.

Date: February 5, 2008 Subject: Enhancements resulting in revision 1.1.52 By: Jeff Kidd

1. Corrected a problem when the computed endpoint of an arc may not join another segment exactly due to rounding. This was causing very small Z up, reposition, and Z down moves. MultiCAM now uses an aperture or tolerance value as specified in the **Pocketing & Profiling Parameters** page to determine if two points are close enough to not require a tool reposition.
2. Allow multiple lines while placing text. In the **Text** input screen, **<Ctrl> <Enter>** places a carriage return in the text area.
3. When the **Edit Paths** option is checked, selection and deletion methods (e.g. **** Key, **Select**, **Unselect** menu options) now operate on Paths segments.
4. Corrected a problem with DXF Import if the user already had MillMaster Pro or TurnMaster Pro installed.