

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 2010

Date: April 21, 2010

Subject: Enhancements resulting in revision 2.0.27

By: Jeff Kidd

1. Retain the zoom and pan drawing limits after moving the *Text*, *Statistics* or *Properties* windows.
2. Add the following functions, used with the scrollable mousewheel, to the *Draw Text* screen
 - <scroll> - change the text spacing (tracking)
 - <Shift> <scroll> - change the text curvature
 - <Ctrl> <scroll> - change the text orientation
 - <Shift> <Ctrl> <scroll> - change the text size
3. The **Tools | Add Bolt Hole Circle** screen now defaults the center of the bolt hole to the current marker position.
4. Corrected problem in the **File | Save As** window. If the file had not previously been saved and the <Cancel> button was pressed, an inappropriate error message was being displayed.
5. Corrected a problem entering data in the *Output CNC File To:* field on the **Options | G-Code Control Setup** screen.
6. Added *Centers of All Selected Circles* checkbox on **Tools | Add Machine Cycle** screen. If checked, a canned cycle will be placed in the center of any selected circles in the drawing.
7. Changed product name to **MultiCam™ 2010**.
8. Improved the handling of certain Windows generated errors when *Placing Text*.
9. Corrected a problem when the user pressed <Esc> on the *Open DXF File* screen. Previously, the user was presented a blank **DXF View/Import** screen.
10. Added *Distance From Center* input field on the **CAM | Pocketing/Profiling Specs** screen. If entered, spiral pocketing will begin this distance away from the center of the pocket.
11. The **Edit | Move Profile** function now moves paths in addition to segments.

12. Default *Tool Material* to **HSS** in **Tool Select and Definition** screen if no value was previously selected.
13. In the **CAM | Create CNC Mill File** screen, generate an **M06 Tx** command (where x = Tool # of the first tool used) if the final tool is different than the first tool used.
14. Added *Auto Place Finish Path* checkbox to the **CAM | Pocketing/Profiling** Specs screen. If checked, a finish path using climb milling will be placed at the end of a **CAM | Rough Cut Pocket** operation.
15. Default the Stock Material to Wrought Aluminum Alloys 5000 6000 7000 Series.
16. Changed the error message on Tool Selection screen if an invalid Stock Material was entered in the Drawing Title Block screen.
17. If placing a finish path on a circle, **MultiCAM** will now choose the best direction (Clockwise or Counter-Clockwise). Previously the finish path segment would always be in the same direction as the circle.
18. Added **Visible?** property to path segments. This feature is accessible via the **CAM | Edit Path/Cycle** screen. For each segment with the visible property set to false, the path segment will not be drawn on the screen, and no edit functions will operate on those path segments. However G-codes will be generated for all path segments regardless whether they are visible or not.
19. Added **Make Visible** and **Make Invisible** functions from the **Edit** menu for paths. These functions will make all selected paths either visible or invisible as selected.
20. Improved the functionality of the **Promote** and **Demote** buttons on the **Order Paths** screen (**CAM | Edit Path/Cycle**). The highlight now follows the previously highlighted field in the listbox.
21. Improved the functionality of the **Next** button applied to **Path Groups** on the **CAM | Edit Path/Cycle** screen. All path segments in the *Prev* or *Next* path group blink instead of just the first or last segment.
22. Improved the **Undo/Redo** functions to include most operations on **Path Segments**.
23. Retain the last used .DXF directory when using the **File | Import DXF** function.
24. Added fields in the **CAM | Edit Path/Cycle** screen specifying where to add new *Path Groups*.

25. Improved the **CAM | Rough Cut Pocket** for ***Lathe Style Pocketing***. When lifting the tool between successive levels, only back out the amount needed to clear the material before repositioning to the next cut location.
26. Corrected a problem with the **CAM | Rough Cut Pocket** menu selection when using *Spiral Pocketing*. If the pocket was contouring an arc, some of the inner paths would create unnecessary path segments.

Date: October 14, 2009

Subject: Enhancements resulting in revision 2.0.8

By: Jeff Kidd

1. Corrected a problem when generating .CNC files. When a tool change was commanded, the corresponding start spindle command (M03) was not being generated correctly.
2. Duplicate the functionality of the <F3> key (*repeat last edit command*) with the <F12> key. On some Vista PCs, the <F3> key runs MS Office Excel.
3. Made the **Tools | Add Machine Cycle** as a valid *repeatable command*.
4. Corrected a problem in the **CAM | Make CNC Mill File** function. If a tool change was immediately followed by a canned cycle (G81 - G89), the canned cycle command was not generated properly.
5. Corrected a problem when the red magnetic snap indicators would inadvertently remain on the screen.
6. Corrected a problem when performing a *Spiral Pocket* on a rectangular cutout. An extra segment was being drawn.
7. Added the ability to export *Tool Setup* parameters via *meta commands*, to be read by **MillMaster 2009**.
8. Corrected a problem with **CAM | Make CNC Mill File** and **CAM | Make CNC Lathe File**. If the tool radius was exactly the same as the radius of an arc in the drawing, a **G02/G03** would sometimes be generated with no parameters.
9. Enhanced the **Edit | Remove Vertex** function for paths to allow removed vertices that are part of arcs. Previously only vertices in line segments were allowed.
10. Enhanced the **Edit | Trim Entity** for paths to extend the path segment as well as trim it.
11. Added *Round Stock* to the **Options | Drawing Block Title** screen.
12. Corrected a problem with *Horizontal* and *Vertical Pocketing*. If a circle was inside the pocket and declared as an *Island*, the pocketing lines would sometimes extend inside the circle.
13. Corrected a problem with *Spiral Pocketing*. In rare cases, due to unusual geometry, a valid pocket existed but no pocketing segments would be drawn.

14. Added **Edit | Join Endpoints w/Arcs** function for both drawing segments and path segments.
15. After importing a *.DXF* file, the **Zoom Extents** function is called to center the drawing in the viewport.
16. Corrected a problem in the **View/Edit Path Segments** screen. If the *Apply Changes to Path Group* option button was pressed, then the **Next** and **Prev** buttons were inadvertently updating Z-values for all path segments in that group.
17. Corrected a problem in the **View/Edit Path Segments** screen. When cycling through the Path Segments, the wrong color was sometimes displaying for the selected segment.
18. Improved the generation of tool path segments. Now fewer zero-length segments, due to rounding, will occur.
19. Added **{Material Shape}:** and **{Diameter}:** meta commands to the **CAM | Make CNC Mill File** function.

Date: May 27, 2009

Subject: Enhancements resulting in revision 2.0.3

By: Jeff Kidd

1. Corrected a problem introduced in Version 2.0.2. The drawing area was losing the zoom level when a menu selection was made.

Date: May 5, 2009

Subject: Enhancements resulting in revision 2.0.2

By: Jeff Kidd

1. Corrected a problem loading .CDM files whose Material description contained a comma character (","). This character was being misinterpreted as a field delimiter causing files saved to be unreadable. This version can read files previously saved.
2. Corrected a problem when resizing the screen or adding/deleting/moving property windows. The drawing area did not always refresh.
3. Updated the Help file. Added a detailed description of all menu items and toolbars. Added an appendix for *Single Line Fonts*.

Date: April 6, 2009

Subject: Enhancements resulting in revision 2.0.1

By: Jeff Kidd

1. Import DXF now defaults to the \DXF directory underneath where **MultiCam** was installed..
2. Corrected a problem on Horizontal or Vertical pocketing. If the pocket contained an arc greater than 180°, sometimes the cuts would not reach the edge of the pocket.
3. Added a one-time "Terms & Conditions" notice.
4. Corrected a problem saving a file on Windows Vista. If the user typed in a filename with no extension an error occurred. Now, if the filename has no extension, the file will be saved as a .CDM file.
5. Changed the .CNC output parameters from 4 decimal places to 5 decimal places.
6. Improved the spiral pocketing logic when the pocket contains arcs. Previously, the pocketing algorithm would sometimes leave a larger than expected empty space in the middle of the pocket.
7. Improved the logic when trying to find **MillMaster Pro** executable file. Now, if no file was specified, **MultiCam** will look for **MillMaster** in "C:\Program Files\Mill Master Pro\".
8. Added a separate federate for Z-Axis plunge. This value can be viewed/changed in the **CAM | Pocketing Profiling Specs** menu selection, and will be defaulted to 1 inch/min.
9. Modified the install package to set the **MultiCAM** directory to be Read/Write for all users.
10. Added a warning message if the Tool Diameter is too big for the pocket selected.
11. Corrected the error "*A property or method call cannot include a reference to a private object.*", when placing text .

Date: October 16, 2008

Subject: Enhancements resulting in revision 1.1.55

By: Jeff Kidd

1. Corrected a compatibility issue with **Windows Vista**. Previously after installing MultiCam the user would receive an "**Unexpected error. Quitting.**" error.
2. Corrected an error in **File | DXF Import**. Occasionally arcs would be drawn inverted (clockwise instead of counter-clockwise).

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.