

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 2013

Date: November 19, 2012

Subject: Enhancements resulting in revision 2.1.65

By: Jeff Kidd

1. Added the ability to do a specified number of Roughing Cuts each time **CAM | Place Finish Path** is selected. The user can enter the *Total Number of Passes* and certain parameters. The default value for *Total Number of Passes* is 1, which will produce only a finishing pass, with no roughing passes.
2. Improved the drawing of the arrow tips on arc segments. On very tight arcs, the angle of the barbs made the arrow hard to see.
3. If the *Step Over Distance* radio button is selected in the **CAM | Pocketing & Profiling Parameters** screen, then recompute the *Step Over Percentage* whenever the current tool diameter is changed.
4. Corrected a problem where **MultiCam** was not appearing as a running application when <Alt> <Tab> was being used to toggle between running apps. This was a problem when the **File | Save As, Options | Printer Options** or the **Tools | Add Bolt Hole Circle** screens were displayed.
5. The **Zoom Window** toolbar icon now behaves as a one-time action. That is, after selecting **Zoom Window** and zooming, the feature is turned off and the cursor is restored to its previous state (i.e. **Select, Unselect, Line, Rectangle, 3-Point Arc, Pan**, etc.)
6. During a **CAM | Place Finish Path**, if a *Lead In/Lead Out* path segment will cause a *Gouge Avoidance* operation, **MultiCam** will advise the user and offer to automatically trim the *Lead In/Lead Out* segments.
7. The Help File title now displays the correct caption reads "*MCFW PDF Help*".
8. In the **Pocketing/Profiling Specs** screen, the *Step Over Amount Percentage* will now accept 6 digits. Previously it only allowed 3 digits. This could have caused a problem if the tool diameter was very small, the *Step Over Amount Percentage* could exceed 3 digits.
9. Corrected a problem with **Horizontal Pocketing**. In some cases, a horizontal cut was missing near an inside vertex of two straight lines.

10. Corrected a problem with **Horizontal Pocketing**. If an arc was present near the bottom of the pocket, the cuts would be drawn incorrectly outside of the pocket.
11. Improved the efficiency of the tool path during **Lathe Style Pocketing**.
12. Added a **<Cancel>** button and **<Use Marker Pos>** buttons to the **Position Cursor** screen.
13. Corrected a problem with the **Entity** details panel on the main form. If a line's properties were changed such that its angle (*Bearing*) was  $> 90^\circ$  and  $\leq 270^\circ$ , then the angle (*Bearing*) would be refreshed with the wrong value ( $- 180^\circ$  of the correct value).
14. If **MultiCam** is loaded by double clicking a **.CDM** file, that file is now logged to the MRU (Most Recently Used) list on the **File** menu.
15. Improved the output of **CAM | Make CNC Lathe File**, such that during "Lathe Style Pocketing", tool back out and reposition moves are done with a **G00 Rapid Move**. Previously, all tool moves were done as **G01** moves.
16. Corrected a problem if the user deleted a path segment in the **CAM | Edit Path Cycle** screen. After the deletion, end markers would inadvertently be displayed on path segments.
17. Improved the functionality of the **Undo** and **Redo** buttons. Previously, some arcs were not working well. Also, the **CAM | Undo Last Place**, and **Pocketing** operations would not **Undo** correctly.
18. Added **Use R-Parameter** for Arcs checkbox in the **Options | G-code Control Setup** screen. If checked, when generating G-code files, G02/G03 statements that are not full circles will use the **R** (Radius) parameter instead of the **I/J/K** parameters.
19. Corrected a problem with **CAM | Place Finish Path**. Sometimes the arcs around obtuse angles would be inserted out of order. This would cause problems with the **Undo** and **Edit Path Cycle** functions.
20. Store the **MRU** (most recently used) file list in a separate file (MCFW.MRU). This prevents any software updates from clearing the **MRU** list.

## MultiCAM 2011

Date: May 17, 2011  
Subject: Enhancements resulting in revision 2.1.48  
By: Jeff Kidd

1. Corrected a problem when **MultiCam** would sometimes fail to generate certain G02/G03 commands when generating CNC Lathe Files (for **TurnMaster Pro**).
2. Added a color palette to the *Tool Select & Definition* screen to allow more than the current 16 possible tool colors.
3. Add the following Meta Commands when generating CNC Lathe Files:  

```
/* {Part Length}: 2.0000 {O.D.}: 1.0000 {I.D.}: 0.0000 *
```

```
/* {Z Origin Zero}:-1 *
```
4. Corrected a problem when unnecessary M06 Tool Change commands were being generated when generating CNC Lathe Files.
5. Lengthened the *Ref Point* display on the **Mirror Segments** and **Rotate Segments** screens.
6. Improved the functionality of the **Edit | Move Profile** feature. Now if any drawing segments are selected, only they will be moved. If no drawing segments are selected then all drawing and path segments will be moved.
7. Corrected a problem where Lead-In and Lead-Out paths were not always being drawn for arcs and circles.
8. Corrected a problem in **CAM | Place Finish Path** and **CAM | Quick Place Selected Segs.** If the first segment was an arc or circle, then the finished path would never draw directly on top of the drawing.
9. Corrected a problem when double clicking a **.CDM** file from Windows Explorer. Sometimes the source file would not load properly.
10. Improved the handling of Z-depth changes on the **CAM | Edit Path Cycle** screen when the *Apply Changes to Path Group* radio button was selected. Now the new Z-depth will only be applied to path segments in that group that had the same initial value. This corrects a problem when multi-depth pocketing groups were inadvertently being set to the same Z-depth.

11. Added File Drag/Drop functionality. If a **.CDM** file is dragged from Windows Explorer to the drawing area of MultiCam, the file will be loaded.
12. Corrected a problem when in *Select* mode or *Unselect* mode. After a **<Ctrl> A** (select all) or **<Ctrl> U** (unselect all) the user was inadvertently being taken out of *Select* or *Unselect* mode. Now the current mode is retained after a **Ctrl> A** or **<Ctrl> U**.
13. Increased the precision of CNC Output when in Metric (Millimeter) mode from 2 to 4 decimal places.
14. Added "Add Fillet" to the Edit Menu for drawing segments. A fillet is rounds off 2 intersecting line segments with an arc of a given radius.
15. Added Drag/Drop functionality for DXF files.
16. Added support for POINT entities in DXF files.

Date: October 27, 2010

Subject: Enhancements resulting in revision 2.0.34

By: Jeff Kidd

1. Corrected **Mirror Point Y** label on the **Mirror Segments** screen.
2. Corrected a problem with generating **Lead-In** and **Lead-Out** path segments on closed drawings. In this case, the **Lead-Out** segment would sometimes be placed incorrectly.
3. Corrected several problems with the **Undo/Redo** functions.
4. If a **DXF Import** was done, remember the name of the DXF file. Use this name as a suggested .CDM file if no file name already used.
5. Corrected a problem when porting finished .CDM files to other PC's. If the path where the file was saved on the original PC does not exist on the new PC an error occurred.
6. When placing a path with a final depth greater than the incremental depth, and the same start and end point, don't lift the tool between passes.
7. Corrected a problem on the Tools screen inadvertently displaying the message "Invalid 'Stock Material' entered in '**Options | Drawing Title Block**' screen".
8. Corrected a calculation error computing the closest point on nearly horizontal or vertical lines. In some cases, a small error (less than 0.0001) may have been returned in either the X or Y values of the closest point.
9. Improved the user interface for entering the type of material in the **Drawing Title Block** screen.
10. Corrected a problem when generating a CNC Mill File. The initial F-parameter (Feedrate) was not always generated.
11. The **F**-Parameter (Feed Rate) will always be after and **K** (Depth per Cycle), **Q** (# of strokes) or **D** (Dwell) parameters on any generated Canned Cycles.
12. Corrected a problem during **Place Finish Path**. Under certain circumstances an unwanted arc would be drawn from the point of the last placed path.
13. Added **Auto-Align to Bottom Left** checkbox to the Import DXF screen. If checked, the drawing will be shifted so that the bottom left of the drawing is at the origin.

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. <Del> 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.