

MecSoft Corporation

18019, Sky Park Circle, Suite KL
Irvine, CA 92614, USA

PHONE: (949) 654-8163
E-MAIL: SALES@MECSOFT.COM
WEBSITE: WWW.MECSOFT.COM

This document outlines the enhancements and the bug fixes that were incorporated into VisualMILL 6.0.4.3 and RhinoCAM 2.0.1.13

VisualMILL 6.0.4.3 & RhinoCAM 2.0.1.13 CAM Enhancements

1. 2 ½ axis engraving – specify the cut type as Zig or Zig/Zag
2. Sorting for 2 ½ Axis Engraving as in Drilling
3. Sorting for V-carving as in Drilling
4. Sorting for Pocketing as in Drilling
5. Add Product Type (STD / STD+4th Axis/PRO) to the about dialog & Registration dialog.
6. List Tools – Users would like to add Spindle RPM & Cut Feed rate to this list.

VisualMILL 6.0.4.3 & RhinoCAM 2.0.1.13 Bugs Fixed

1. 2 ½ axis engraving – Arc is being fit in a plane that is not in XY XZ or YZ. This causes problems when code is posted.
2. Some areas are not machined by Horizontal Finishing under certain conditions
3. Optimize setting in Horizontal Finishing does not work under certain conditions.
4. Tool center programming – Creating a toolpath using Ball End Mill offsets the toolpath in Z by the tool diameter instead of tool radius.
5. 3-Axis Radial Machining – Start point resets to outside when the cut traversal is set to Zig.
6. Work offset Prefix – always outputs the letter ‘G’ with Set MCS operations. Set MCS is not reading fixture offset prefix variable from post processor located under Misc Tab.
7. Cut ordering for 2 ½ axis pocketing, facing & all 3-axis operations machines regions in reverse order picked.
8. 4th axis Rotate table always outputs Feedrate value of 1.
9. Shop documentation (Template 2) loses the sequence of screen shots when a Set MCS or Rotate table mop is part of a mop set.
10. Coolant Through does not get posted (only Flood & Mist work).
11. Mops info lists the subtotal in reverse order when you have multiple Mop Sets and sequence of mops are not in order.
12. Z level toolpath display does not work at all time.
13. Renaming a Mop in Vista, switch to simulate tab and click simulate, the Mop name adds additional characters.
14. Extraneous tool holders are being displayed when a tool change occurs when using multiple viewports.
15. Setup tab – Machine Setup – Set this to 4 or 5 Axis. Now edit the machine setup dialog and click cancel. The display in the browser switches to 3 axis. When you edit the machine setup dialog the machine type shows up correctly.
16. Machine Setup dialog – when user selects the 5 axis or 4th axis as machine type and clicks specify for the primary axis the X and Y values does not reset to 0.
17. RolandCAMM-GL Output –
 - a. Plunge feed-rate value always posts as 0

- b. 4th axis rotate table – Retract to tool change point before each table rotate operation does not output the Z value. Only XY values are being posted.
18. 3D offset profiling –
- c. Use the start point of curve for start of entry/exit.
 - d. The toolpath cuts from bottom and works its way to the top. There is no automatic way to switch the direction.
19. When the user clears out all the values in a dialog to enter feeds/speeds, the error message pops up – “Please enter a number” twice.
20. Mirror the toolpath in YZ plane and arcs are reversed.

VisualMILL 6.0.4.3 Bugs Fixed

1. V-carve toolpath issue for certain customer files
2. "Parts offset stock", and move the stock/parts origin, then VisualMill crashes.
3. Cannot create a User Defined tool in VisualMILL 6.0 if model is in mm

RhinoCAM 2.0.1.13 Bugs Fixed

1. Creating a user defined tool crashes RC 2.0 when Rhino is set to Millimeters.
2. Pencil trace crashes RhinoCAM with a Flat End Mill
3. Pencil trace crashes RhinoCAM with a Corner Radius Mill
4. Pencil Trace Crashes under certain geometry conditions
5. Parallel Finish Crash under certain geometry conditions
6. Rhino crashes when trying to open a specific user file
7. Deleting layout crashes Rhino when RhinoCAM is loaded.
8. Unable to load Tool Library at startup