

# 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 6.0.5.10 and RhinoCAM 2.0.2.10

## **VisualMILL 6.0.5.10 & RhinoCAM 2.0.2.10 CAM Enhancements**

1. Insert multiple lines of comments under the Create/Edit tool dialog
2. Installer will not wipe out user settings when installing/uninstalling
3. In the feature selection dialog when a curve is selected in the list box the corresponding curve on the graphics screen will be highlighted.
4. Knowledge bases loading/saving consolidated in Mops toolbar
5. Additional Mill posts have been added

## **VisualMILL 6.0.5.10 & RhinoCAM 2.0.2.10 Bugs Fixed**

1. Select a circle & create a chamfer toolpath with a V-Mill. Cut Start Side - Outside. 2d entry/exit gouges the geometry.
2. Select a circle & create a 2 ½ axis Profiling with a 0.5” Flat End Mill. Set Stock = -0.25, Cut Start Side – Outside. 2d entry/exit gouges the geometry.
3. V-Carving – Cut Transfer- Skim clearance parameter is ignored. The tool always retracts to Clearance Plane.
4. Hole Pocketing- the spiral cuts are backwards
5. Knowledge bases loading creates hidden and multiple Mops

## **VisualMILL 6.0.5.10 Bugs Fixed**

1. Cannot create a user defined tool (both inch & mm) – Invalid tool definition error appears when you click save as new tool.
2. Solid Extrude curve & Surface Extrude curve fails and errors out “Non planar Geometry encountered when 2 curves are extruded”.
3. Intersection Point Snap does not work in some cases.

## **RhinoCAM 2.0.2.10 Bugs Fixed**

1. Faceting bug when toolpath is regenerated.
2. 2 ½ Axis Hole Pocketing – Spirals are not being generated when there is room for the tool to move around.