What's New in RhinoCAM 2016

This document describes new features and enhancements introduced in RhinoCAM 2016, the CAM plug-in for Rhino from MecSoft Corporation.
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>RhinoCAM 2016</td>
<td>3</td>
</tr>
<tr>
<td>What’s new in RhinoCAM 2016</td>
<td>3</td>
</tr>
<tr>
<td>What’s new in RhinoCAM 2016 - MILL</td>
<td>3</td>
</tr>
<tr>
<td>2-Axis Enhancements</td>
<td>3</td>
</tr>
<tr>
<td>3-Axis Enhancements</td>
<td>7</td>
</tr>
<tr>
<td>4 Axis Toolpath Enhancements</td>
<td>9</td>
</tr>
<tr>
<td>Drilling Toolpath Enhancements</td>
<td>9</td>
</tr>
<tr>
<td>UI Enhancements</td>
<td>9</td>
</tr>
<tr>
<td>Feeds/Speeds Enhancements</td>
<td>9</td>
</tr>
<tr>
<td>Machining Regions Enhancement</td>
<td>9</td>
</tr>
<tr>
<td>Simulation Enhancements</td>
<td>10</td>
</tr>
<tr>
<td>CAM Utilities</td>
<td>10</td>
</tr>
<tr>
<td>Miscellaneous</td>
<td>10</td>
</tr>
<tr>
<td>What’s new in RhinoCAM 2016 - TURN</td>
<td>11</td>
</tr>
<tr>
<td>Common Enhancements to RhinoCAM 2016 MILL &amp; TURN Modules</td>
<td>12</td>
</tr>
<tr>
<td>Tooling Enhancements</td>
<td>12</td>
</tr>
<tr>
<td>Material Removal Simulation Enhancements</td>
<td>12</td>
</tr>
<tr>
<td>Knowledge Base Enhancements</td>
<td>12</td>
</tr>
<tr>
<td>What’s new in RhinoCAM 2016 - NEST</td>
<td>13</td>
</tr>
<tr>
<td>What’s new in RhinoCAM 2016 - ART</td>
<td>13</td>
</tr>
</tbody>
</table>
What’s New in RhinoCAM 2016

RhinoCAM 2016

This document describes the new functionality that has been introduced with the release of the RhinoCAM 2016 product. The enhancements incorporated into each of the constituent modules of RhinoCAM are also described in detail.

RhinoCAM 2016 is a plug-in that runs inside the Rhinoceros 5.0 CAD modeler and hosts the following modules:

1. RhinoCAM-MILL
2. RhinoCAM-TURN
3. RhinoCAM-NEST
4. RhinoCAM-ART

Each of these modules can be licensed and invoked separately of the other modules.

What’s new in RhinoCAM 2016

This section describes the common enhancements and changes to RhinoCAM 2016, which is the base platform that hosts the other CAM modules such as MILL, TURN and NEST.

1) RhinoCAM has been transitioned over to use software licensing from a hardware key (dongle) licensing scheme used in previous versions.
   a. Upgrade licenses will require the dongle to be plugged in the first time the product is upgraded but will not be necessary after the upgrade process completes
   b. Internet access is necessary when activating the license for the first time. Once activation takes place, internet access is not necessary for normal operation

2) All old style dialog pictures and icons have been reworked for a more modern look

3) On-line help for all modules completely rewritten and enhanced

4) Cut material simulation libraries have been upgraded and enhanced for better quality and performance

What’s new in RhinoCAM 2016 - MILL

This section describes the enhancements and changes to the RhinoCAM-MILL 2016 module.

2-Axis Enhancements

1. Use of Stock model to generate the material regions instead of having to select material regions in 2-1/2 Axis milling. This has the advantage of not having to create 3D part models if you are working only with curves. This has the added advantage of being able to machine arbitrary open area machining. A new method called 2-1/2 Axis
Roughing has been introduced to implement this. The heights of the selected regions are honored in this method.

2. **Overlap Distance for Closed Profiles Added.** The Overlap option on the Exit block in the Profile Engage/Retract page has been added. Specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.
3. Option for Triangular or Rectangular bridges added for profile machining
4. Zig Zag option for Z levels in profiling allowing for less retracts between Z levels.

5. Use overlapping Stock and Part regions for machining open pockets. Not necessary to create open regions to define open pockets in 2-1/2 Axis Facing.

6. Arc fitting of toolpaths automatically turned on now.
3-Axis Enhancements

1. Horizontal Roughing has been enhanced to add Spiral and Radial cut patterns for machining each cut level. The dialogs also have been unified with 2-1/2 Axis Pocketing making this cut method even more easy to use.

   Spiral Machining in each cut level  
   Radial Machining in each cut level

2. Corner cleanup now implemented in Horizontal Roughing

   Inside Corner Cleanup loops in each level

3. Cutting Top Only or Top And Sides control added to Parallel Finishing in 3 Axis machining

   Top Only Machining  
   Top & Sides Machining
4. Added an option to cap holes in Parallel Finishing in 3 Axis machining

![2015 - Tool drops into hole](image1)

![2016 - Tool ignores hole if Ignore Holes on](image2)

5. Horizontal Finishing - Reduce retracts in optimized machining

![2015](image3)

![2016](image4)

6. Arc Fitting property page added in Horizontal Finishing as well as Parallel Finishing
4 Axis Toolpath Enhancements

Machining from rectangular stock in 2015

Same toolpath in 2016

1. R-Level Roughing has been completely revamped to produce better toolpaths
2. R-Level Finishing has been completely revamped to produce better toolpaths as well

Drilling Toolpath Enhancements

1. Pick Z level has been added to all hole making operations

UI Enhancements

1. Moved all CAM options and preferences into one dialog
2. Dialog behavior in dual monitor setup enhanced.
3. Machining Operations Info. - Subtotals for each mop set are now provided when a setup or machining job includes multiple mop sets
4. Now allowing a way to disable automatic loading default library in the Tools Browser
5. 4 Axis dialogs enhanced to show containment graphically on the screen
6. Orient part in RhinoCAM now has ability to orient part normal to active C-Plane as well as to move a point on a surface/planar curves/pick point.

Feeds/Speeds Enhancements

1. The system now allows coolant specification in feeds/speeds dialog in the operation to override the one in tool.
2. Editing a mop by replacing the current tool with a new one does not update feeds/speeds. This has been implemented.
3. Updating feeds/speeds on a tool & saving edits to tool, requires feeds/speeds on each mop to be updated by editing a mop & selecting load from tool from feeds/speeds tab. This is now done automatically.

Machining Regions Enhancement

1. Display each contiguous region as sub object of the Machining Region object. Users should be able to edit these sub-objects independently
Simulation Enhancements

1. The previously separately priced Advanced Simulation Module is now included in the Standard configuration of the Mill module.
2. It is now possible to perform view manipulations such as rotating and moving of the graphics screen for better visualization when the Part/Stock dialog is active.
3. Simulation of Instancing operation has been changed to simulation of all instanced operations instead of just the first one as was done in previous releases.

CAM Utilities

1. A new utility called “Explode Cabinet Design” has been introduced. This is useful for wood cabinet makers to explode a cabinet design from cabinet design software into its constituent flat panels and lay them out for machining.

Miscellaneous

Numerous other smaller usability enhancements and over 200 bug-fixes....
What’s New in RhinoCAM 2016

What’s new in RhinoCAM 2016 - TURN
This section describes the enhancements and changes specific to the RhinoCAM-TURN 2016 module.

1) Display issues when displaying Part and Stock model has been fixed
2) The tool preview display in the Tool Definition dialog now displays differently depending on Tool Tip or Tool Center programming set in the Machining Preferences
3) Pull out variable for threading added to post processor
4) “Always create thread with no taper” check box added to threading parameters dialog. This allows users to pick points on a non-straight area of the model and create a straight thread with no additional geometry creation.

![Parameter to force straight threads](image)

Effect on thread computed point
Common Enhancements to RhinoCAM 2016 MILL & TURN Modules

This section describes the enhancements and changes that are common to the MILL and TURN modules of RhinoCAM 2016.

Tooling Enhancements

1) Tool list display in the Tool Definition dialog now follows the same sorting rules as the tool display in the browser window.

Material Removal Simulation Enhancements

1) Simulation libraries in Milling and Turning have been upgraded and enhanced with:
   a. Multi-threaded simulation
   b. Faster simulation
   c. Smaller memory footprint

Knowledge Base Enhancements:

1) Add the ability to save Drive/Containment geometry selection rules & filters in Knowledge Bases has been introduced. This allows Knowledge Bases, upon load to automatically, select geometry into the operations. This functionality greatly enhances the use of Knowledge Bases in machining automation.
What’s New in RhinoCAM 2016

What’s new in RhinoCAM 2016 - NEST
This section describes the enhancements and changes to the RhinoCAM 2016 NEST module.

1. Dot annotation tagging has been introduced
2. We now allow preview of tags during preview of the nesting
3. Maintaining color properties when parts are nested to sheet now
4. Maintaining group properties of curves in Rhino when parts are nested to sheet now
5. To be able to use nesting for engraving & sign making, we would need the ability to nest curves inside a hole as shown below. So the rectangle would be the part, the exterior & interior of the letters would have to be treated as holes. This has been implemented.

6. We now allow holes or children to be partially inside the part and not enforce complete enclosure in the part. This situation happens in cabinet work when grooves sometimes stick out of the parts.
7. We now have the ability to create the sheets and nested parts in the original layer. This means each sheet will NOT be created in a new layer. The original layer, group and color properties of the parts and sheets will be maintained.
8. We now have the ability to create the sheets and nested parts in separate groups. If this item is selected, each output nested sheet will be added to a separate group using the same naming conventions used for the layers.
9. A new option to arrange the sheets along X, along Y or Stack as we do now has been implemented. Stack will be disabled when user selects not to create separate layers for each sheet in 1 above. All three options will be enabled when the user elects to create each sheet in a separate layer or group.

What’s new in RhinoCAM 2016 - ART
This RhinoCAM 2016 - ART module was improved with numerous small bug fixes. No major enhancements were implemented.