



What's New in VisualCAD/CAM 2024

Feb 6,
2024

This document describes new features and enhancements introduced in VisualCAD/CAM 2024.

CONTENTS

VisualCAD Enhancements	3
Common VisualCAM Enhancements	3
Licensing Enhancements	4
What's new in the MILL module.....	5
Usability Enhancements	5
Fixtures Enhancements	6
Hole Making Enhancements.....	7
2-Axis Enhancements	8
3-Axis Enhancements	10
4-Axis Enhancements	14
5 Axis Toolpath Enhancements	16
Simulation Enhancements.....	18
Machine Tool Simulation Enhancements.....	18
Post Processor Enhancements	18
Bugs Fixed	18

What's New in VisualCAD/CAM 2024

This document describes the changes and new functionality that is being introduced with the release of the VisualCAD/CAM 2024 product. We have made several improvements to the product, including fixing a significant number of bugs and improving performance. We believe that these changes will enhance your experience and make the product more user-friendly. We are committed to providing a high-quality product and hope that these improvements will help to build your trust and satisfaction with our product.

This document is organized by listing and describing each of the enhancements incorporated into the constituent modules of VisualCAD/CAM.

VISUALCAD ENHANCEMENTS

The following improvements have been made to VisualCAD 2024.

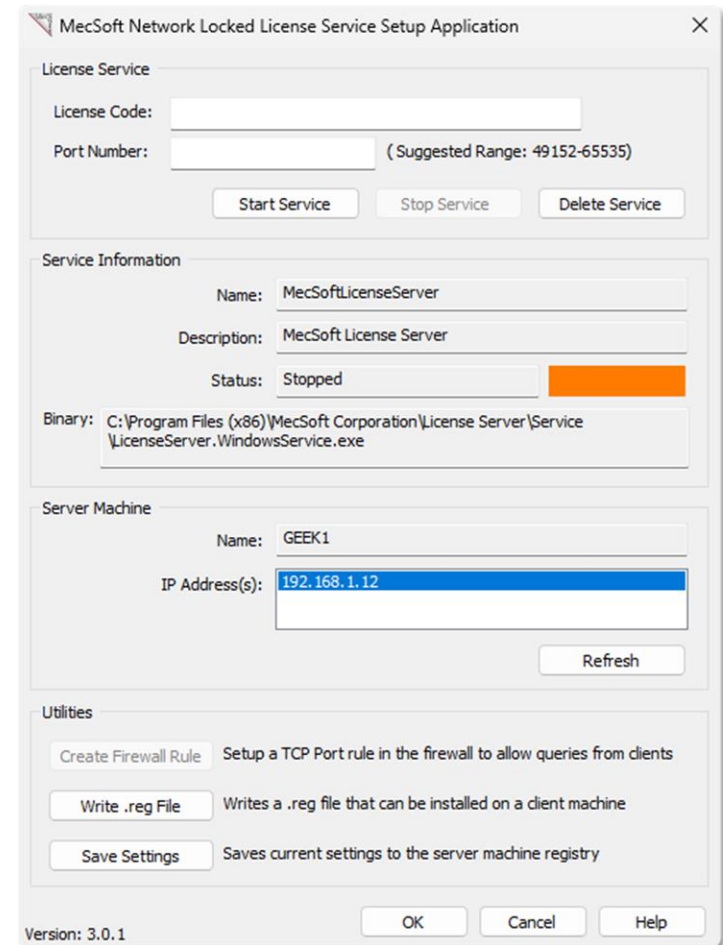
1. Import/Export of DWG files was improved. Unicode Layer names are now correctly handled in these translators
2. Updated windowing system libraries were incorporated in this version of VisualCAD
3. Construction commands were made more robust by code refactoring.

COMMON VISUALCAM ENHANCEMENTS

1. New simulation libraries from Machineworks Inc. have been incorporated into all machining modules. These libraries have improved performance significantly as well as fixed many reported issues.

LICENSING ENHANCEMENTS

1. A new user interface for the Network Locked license model has been introduced along with additional tools for license managers to make it easy for them to manage licenses.
2. The Check for Updates has been made more robust and an error logging system has been introduced to help troubleshoot issues with the deployment of the product.

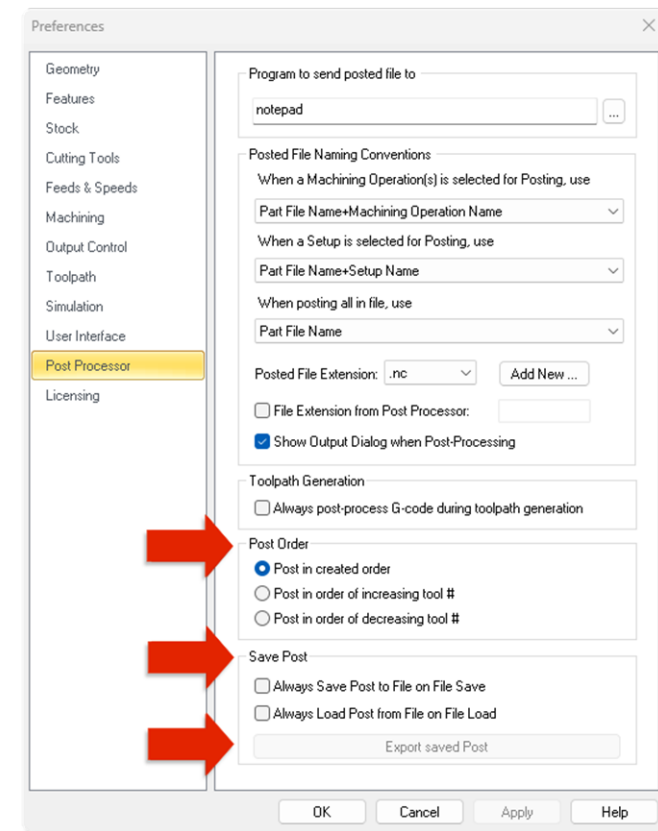


WHAT'S NEW IN THE MILL MODULE

This section describes the enhancements and changes to the MILL module.

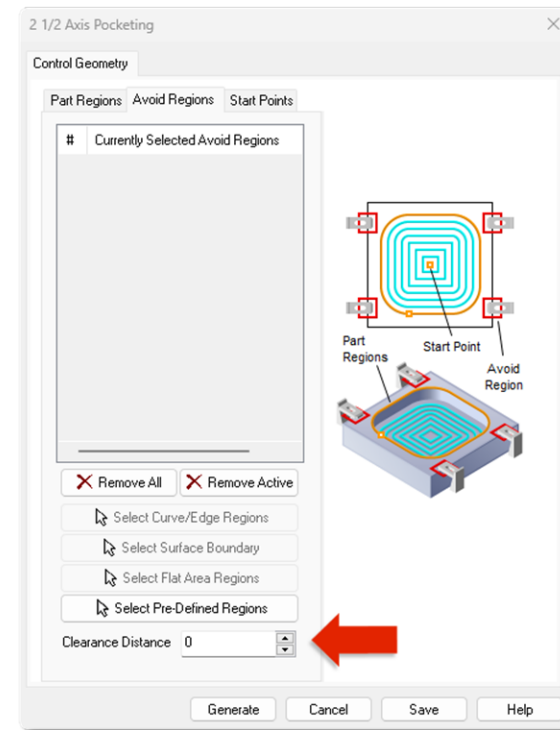
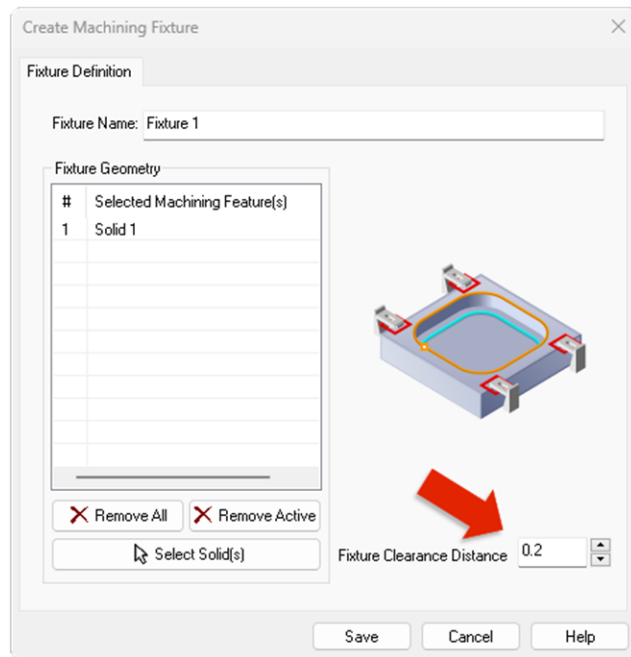
USABILITY ENHANCEMENTS

1. Enhancement to post toolpaths based on tool priority was implemented. Users can now select multiple operations and post them in order of the tools being used, rather than the created order.
2. Ability to save and load Posts in file was separated out from saving and loading Settings.
3. A button to "Export saved Post" was implemented. This allows users to export posts saved in part files. This button will be enabled only when a post-processor is saved in a part file.
4. A "Do not show" again option was implemented in multiple dialogs to enhance usability of the product.
5. All internal code was refactored and restructured to enable additional robustness and speed improvements.



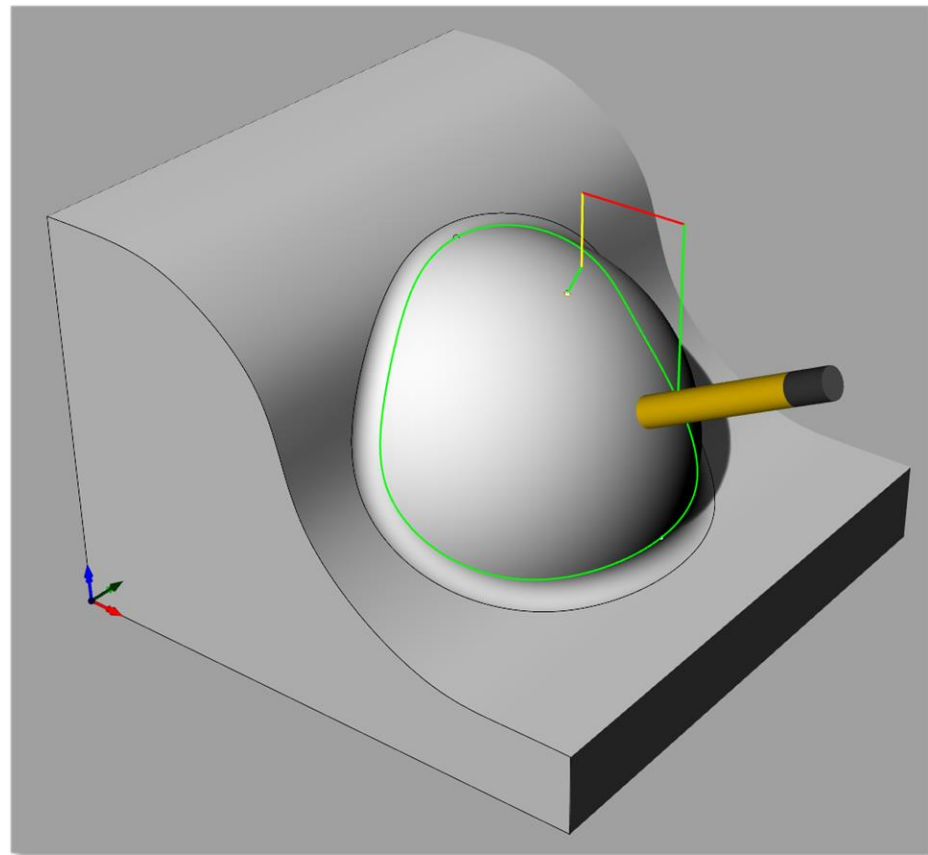
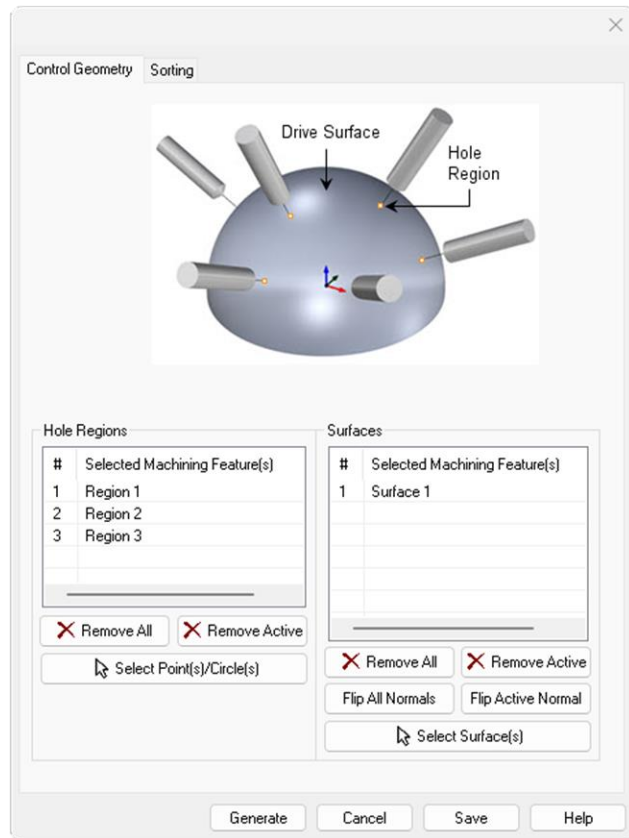
FIXTURES ENHANCEMENTS

1. Fixture Clearance parameter was implemented. Clearance values now can be assigned to clearance geometry that will allow the cutter to stay away from features by this specified amount. A clearance distance can also be specified in the Avoid Region selection dialog in 2 1/2 Axis Operations as well.



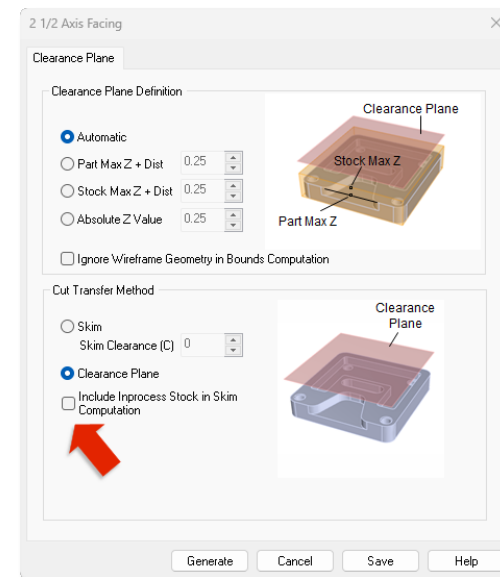
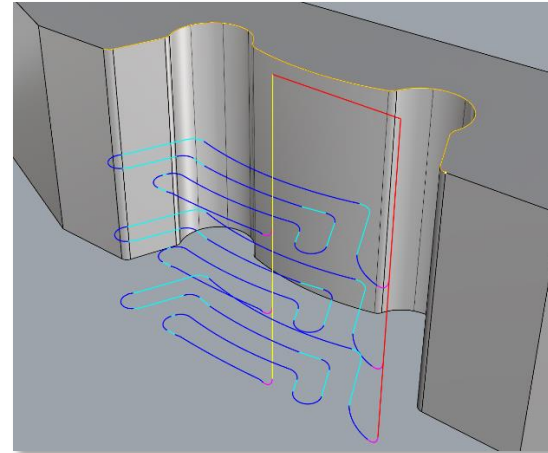
HOLE MAKING ENHANCEMENTS

1. 5 Axis Drilling using drive surfaces was implemented. This method allows the user to select drive surfaces to project drill geometry to and move between drill points using 5 axis continuous movements.

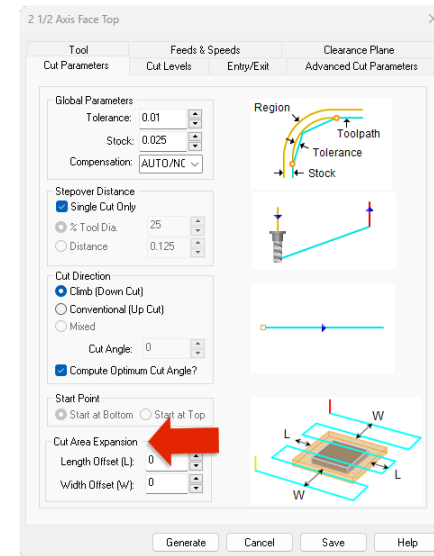


2-AXIS ENHANCEMENTS

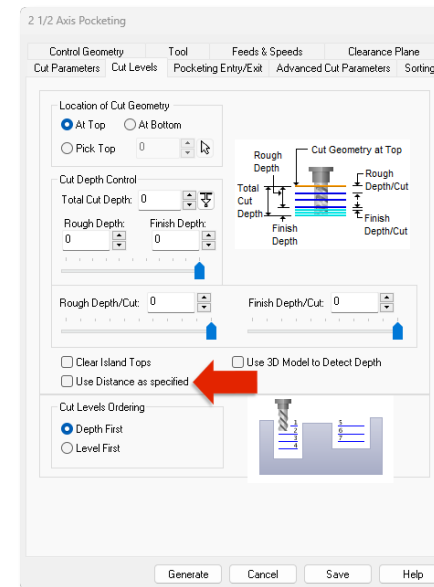
1. High Speed machining in Facing operation was implemented.
An example of the high-speed toolpath is shown here.
2. Skim clearance computation now can account for in process stock in pocketing and facing.



3. Negative offset values for expanding the cut area in the Cut Area Expansion parameters in Face Top Machining was implemented. Users can now specify negative values to shrink the area to be faced.



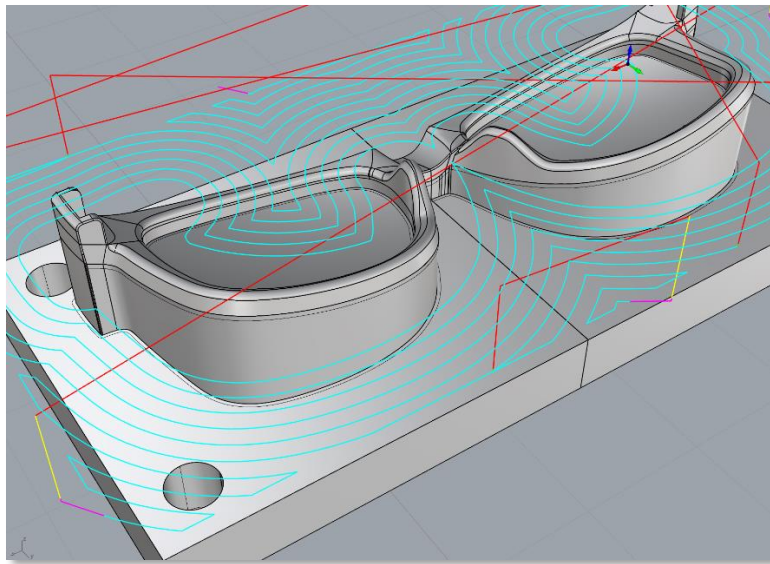
4. Exact Z level creation has been added as an option in Pocketing, Profiling and Facing operations.



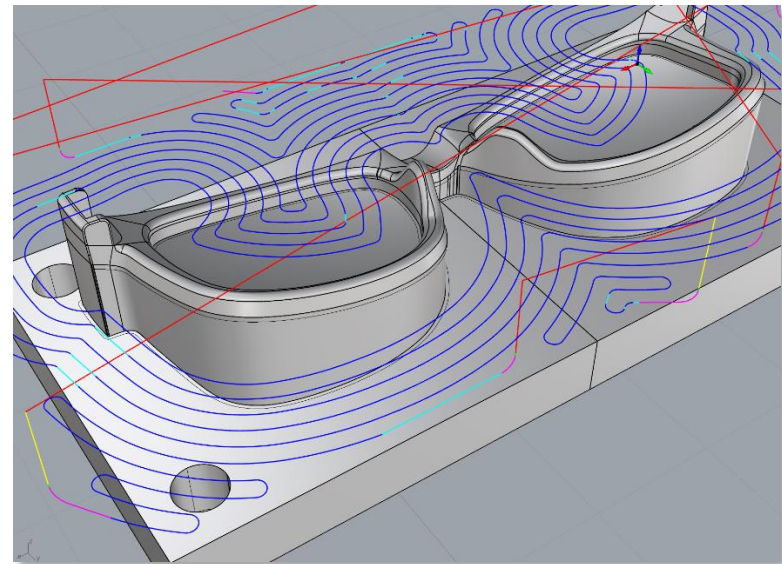
3-AXIS ENHANCEMENTS

1. A high-speed machining toolpath in Facing has been implemented. This method adds 2D and 3D arcs during cut connections as well as removes sharp edges from toolpath motions.

An example contrasting the standard facing operation against the high-speed facing option is shown below.



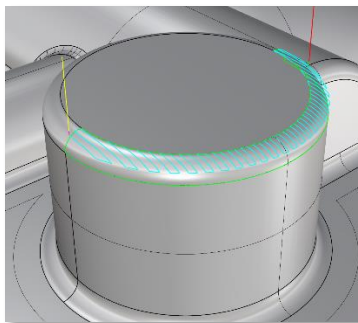
Island Offset Toolpath Method



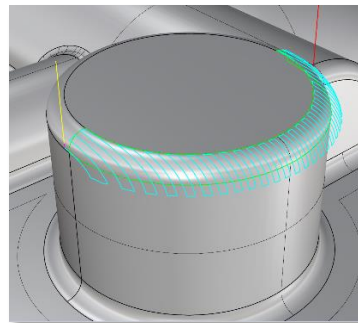
High Speed Toolpath Method

2. Tool CONTACT condition for Part Surfaces has been implemented in 3 axis finishing machining methods such as Parallel Finishing and Horizontal Finishing.

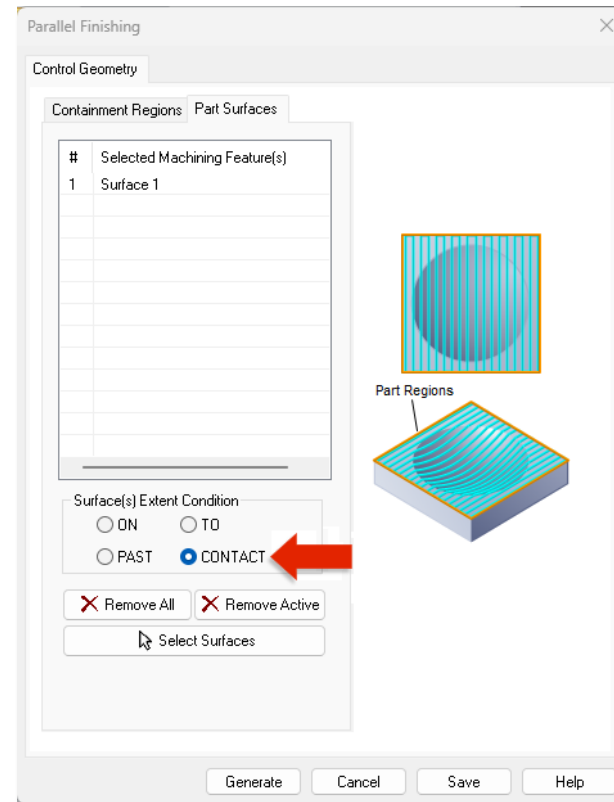
The contact condition is used to start cutting when the tool starts contacting the surfaces being machined and stops when the tool loses contact with the surfaces. An example with the difference between the To and the Contact condition toolpaths is shown below. This can be very useful when machining mold surfaces.



ON Condition



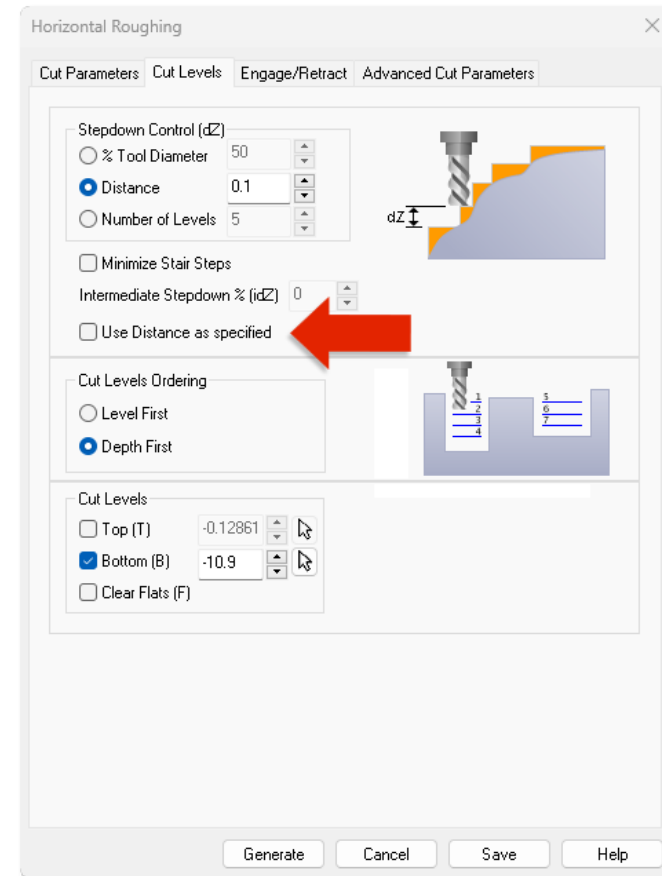
CONTACT Condition



Contact Condition Selection Dialog

3. Exact Z level creation has been added as an option in Horizontal Roughing and Finishing.
In previous releases, the user specified distance between successive Z levels was considered as the

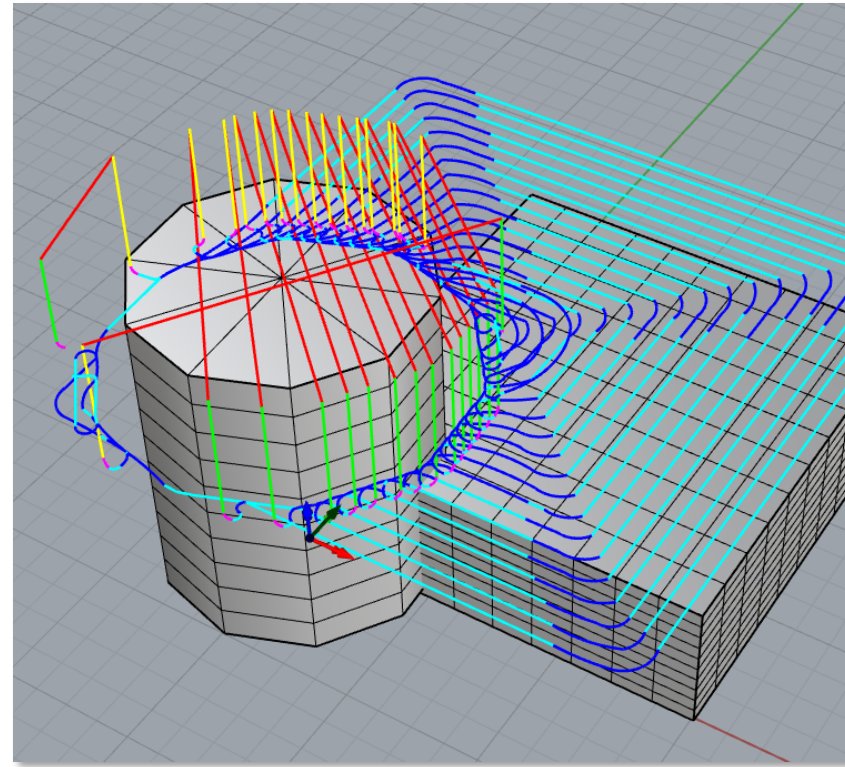
maximum amount of step down possible with the tool. The distances between Z levels were adjusted to yield an integral number of cuts and yet to stay below this user specified maximum step-down value. However, some applications need to have the Z levels to be exactly as specified by the user. Hence this parameter was introduced to handle such applications.



Exact Z Level Option

4. Clear Flats Machining in Horizontal Roughing has been made more robust, to be able to handle large terrain models with numerous flat areas to be machined. An option to not check Clear Flats Z levels against compute Horizontal Roughing Z levels to remove duplicates was also implemented.

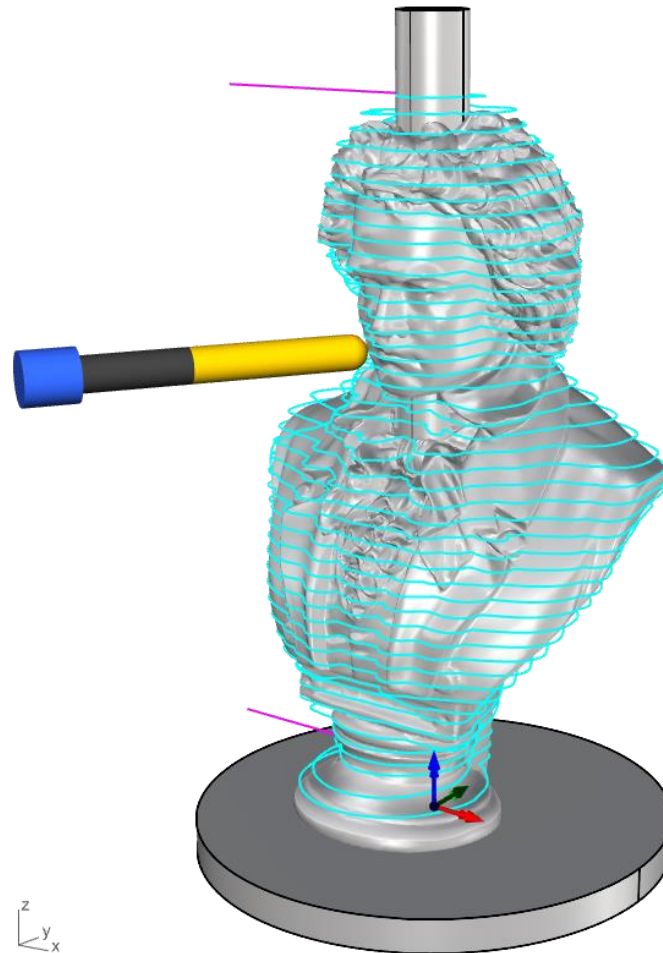
5. Skim clearance in roughing to account for in-process stock for Horizontal Roughing was implemented. In previous versions only the part geometry was considered for skim clearance computations. This resulted in the tool colliding with un-cut stock in some situations. In 2024, the in-process stock model is also used in the skim clearance computations.
6. Meshes can now be selected as the drive surfaces in the Adaptive Roughing. In previous versions only surfaces were able to be selected.



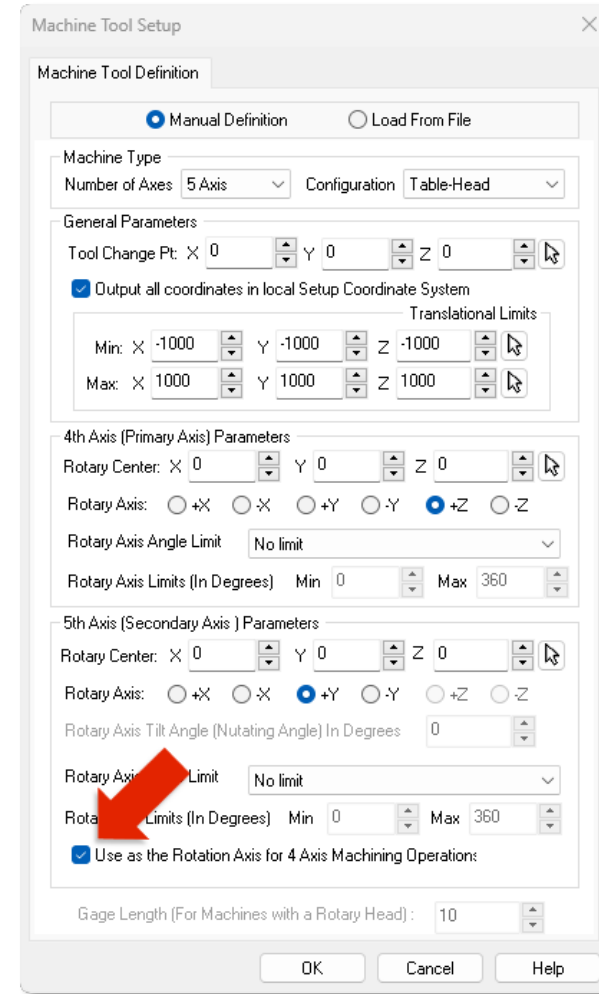
Mesh Selected as Part Geometry

4-AXIS ENHANCEMENTS

1. The Z axis can now be used as a rotational axis in 4 Axis machining operations. If the user selects the Z axis as the primary axis of rotation in the machine tool definition dialog, we now allow the primary axis to be used as the rotational axis in 4 axis operations. This facilitates “tomb stone machining” when parts are fixtured vertically. A simple example is shown here.



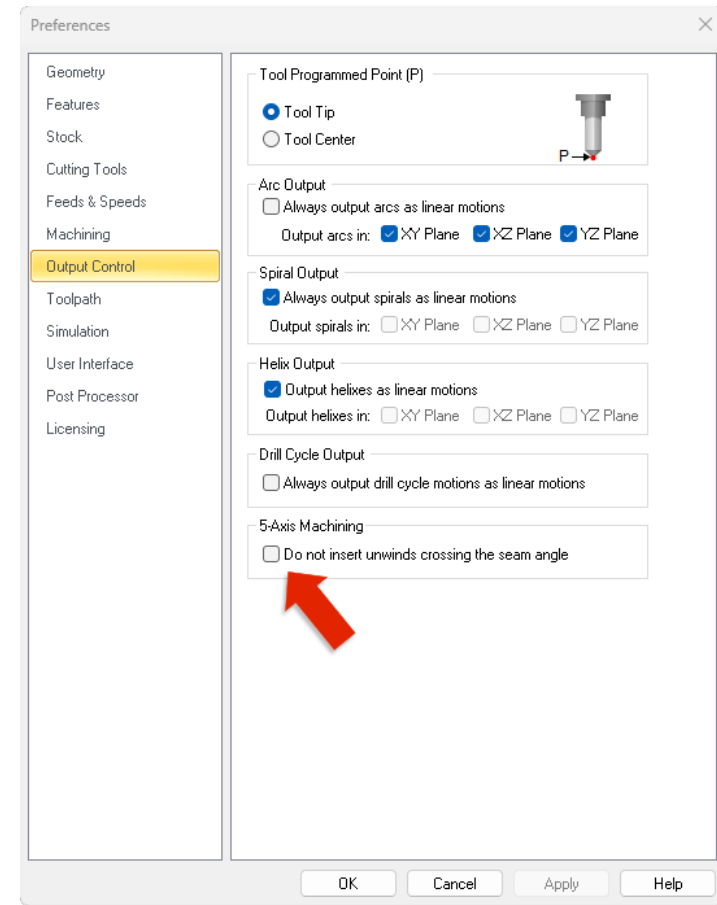
- The Secondary axis of a 5 axis Machine tool can now be used as the rotation axis for 4 Axis Milling. This eliminates the need to have 2 different posts, one for 4 axis machining operations and one for 5 axis machining operations.



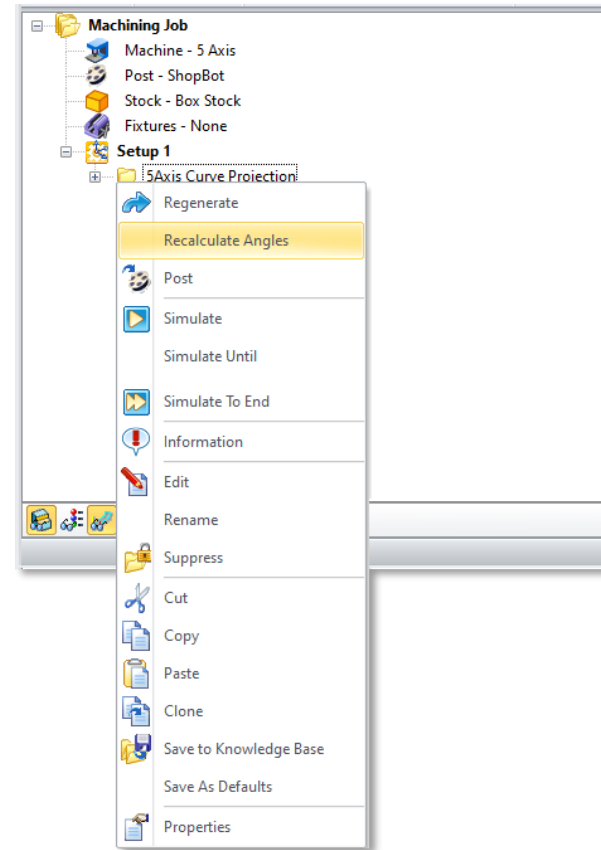
- 4 Axis continuous toolpath computations for Local Coordinate output has been enhanced and made more robust

5 AXIS TOOLPATH ENHANCEMENTS

1. A parameter to *not* automatically insert an unwind operation when crossing the was implemented in the Output Control Options dialog.



2. A separate command to recompute all machine angles based on the order of the operations was added to the Right Mouse Button menu in the Machining Browser. This command recomputes only the 5 axis rotary angles based on the machine kinematics and does not recompute the operations themselves. This can help when users reorder operations in the Job tree.



3. New toolpath generation libraries for 5 axis machining have been integrated with the 2024 products. These libraries implement bug-fixes and performance improvements in 5 Axis continuous machining.
4. Gage Length computation was turned off, if "Output in Local Coordinate System" is checked in the Machine tool definition dialog.

What's New in VisualCAD/CAM 2024

SIMULATION ENHANCEMENTS

1. Fixtures geometry can now be displayed when performing machine tool simulation.
2. New Machineworks simulation libraries have been integrated with the 2024 product. These libraries fix various bugs as well as implement some performance optimizations.

MACHINE TOOL SIMULATION ENHANCEMENTS

1. Additional machine tool models have been added as part of the installed machine tool simulation library.

POST PROCESSOR ENHANCEMENTS

1. Allow setting of both inches and mm codes at the same time in the Post Generator without switching to corresponding units was implemented. Now the post-processor will automatically output the correct output based on the units of the programmed part. It will also scale the corresponding stored values for feed rates. This negates the need to have two posts for every machine.
2. Post variables for laser tool specific parameters such as Pierce, Dwell, and Power were added.
3. Enhancement to add air coolant thru the spindle was implemented.
4. Output Setup Code can now be optionally inserted before Tool Change Code.
5. Ability to control the location where Tool list is output in the posted code was implemented.
6. Variables for outputting each operations machining time was implemented.
7. Variables to output comment start and end characters have been introduced.

BUGS FIXED

Numerous bugs have been fixed to make the product more reliable, robust and user friendly.

1. Issues with CSV export for Tool libraries when tested in CAM 2020 were fixed.
2. CAM crashes when start regeneration during simulation process was fixed.
3. The cut motion start code not being output was fixed.

4. Cannot clear Program # under Machining Operation Properties dialog once you enter and click OK was fixed.
5. Custom tool incorrectly defined when a tool holder is assigned to it was fixed.
6. CAM crashes when opening Exported VCP file from Rhino was fixed.
7. Fixed issue with rotation angle output for setups even if setup block code is empty.
8. Using pick to determine min & max diameter for diameter filter range in hole making does not work when face edges are selected was fixed.
9. Serious unhandled error when simulation reaches a Machine Control operation.
10. Fixed nesting window freezing issue while auto tag in geometry side.
11. Fixed crashes in the Nest module.
12. Clearance value updated for nesting shapes.
13. NEST returns error "Available orientations for part(s) are not suitable for Nesting" and fails to nest all parts.
14. Un-suppressing machining operation resets count and priority under Nesting parameters tab in Profile-Nesting to default values was fixed.
15. Bugs in Explode Cabinet Parameters dialog was fixed.
16. Cabinet Decomposition to Flat Panel Layout was renamed.
17. 5 axis tool axis control fails to post the proper C axis angles in all tool motions was fixed.
18. Software freezes and does not respond on posting 4 axis Machining Operation was fixed.
19. Fixed minor bugs in the CAM installers.
20. Allow negative values for the Width Offset value was implemented.
21. While selecting Select Flat Area Regions as containment region
22. Selection of Edged
23. Cycle off code does not output in the post for 2024 product was fixed.
24. CAM freezes and does not respond on posting 5 axis MOp was fixed.
25. Added code for replacing characters ('/')
26. Tool load block not output for programmable post was fixed.
27. Tool change block not output for programmable post processor was fixed.

28. Post output incorrect angle for first setup rotation for 4/5-axis operations was fixed.
29. Update translator plugins to use MSVC 2019 toolset.
30. VisualCAD/CAM crashes when opening Exported VCP file from Rhino.
31. Explode Cabinet Parameters UI dialog errors were fixed.
32. Diameter range filter not working for Knowledge base selection rules issue was fixed.
33. CAM crashes while saving the "Knowledge Base" from Machining Objects Browser was fixed.
34. Crash when switching from Nesting to Milling was fixed.
35. Crash after clicking "Yes" in the Voxel Simulation dialog and opening a new part was fixed.
36. "When copying a Drill along a curve operation, the parameters are reset to Select Hole Features" was fixed.
37. "When using fixtures, the 2 ½ Axis Facing toolpath won't calculate" was fixed.
38. "Toolpath will show as edited after the warning appears" was fixed.
39. "When producing holes, the "Always post-process G-code during toolpath generation" results in an error" was fixed.
40. "In CSV files the '# of Flutes' heading appears twice, both for drills and mills" was fixed.
41. "Choose Nesting Parameters, tag options and nested sheets layout options exclude each other." was fixed.
42. "Scroll buttons are not working for Machining Objects dialog." was fixed.
43. "Update default settings for User Interface Preferences" was fixed
44. "Changing the CAM modules to G-code editor/MESH some unknown text appears in the Mill module Mops (While using auto hide option)" was fixed.
45. "Auto hiding the Machining Browser hides the CAM module check mark in the CAM dropdown" was fixed.
46. "Machining browser toggle button to change the CAM modules are not working while auto hiding the Machining browser." was fixed.
47. "When copying or cloning a "drilling along a curve" toolpath, the parameters are reset to "Select Hole Features"" was fixed.
48. Fixed issue with draggable command icons on tabs in the Machining Objects dialog
49. "Options under the VisualCAM Machining-Browser/Machining Objects are locked (inability to click) after using Regenerate All operation." was fixed
50. Fixed issue with Programmable post-processor outputting code specified on OnMOpStart at the OnMOpEnd event.
51. Fix for crash when "Enter" key is pressed in the fixture creation dialog
52. Saw machining changed so that the Saw axis is parallel to the X axis of the setup.
53. Fix for Programmable post: multiple lines merged into one when no prefix letter is specified.

What's New in VisualCAD/CAM 2024

- 54. Fix for crash while saving the "Knowledge Base" from Machining Objects Browser.
- 55. Network Locked Server changes to fix crashes due to bad requests.