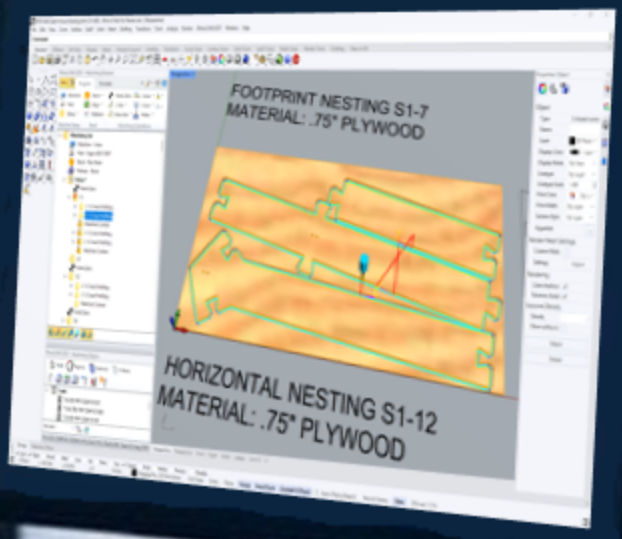
MecSoft

2026 CAMJam



Includes:

- CAMJam 2026 Redesign
- What's New in 2026
- Configurations & Modules
- 2.5 Axis Support Issues
- New 3, 4 & 5 Axis Videos
- System Performance Updates!
- Mastering VisualCAD
- Indexed Machining
- Real-World RhinoCAM Demos
- TURN, NEST, Profile-NEST, G-Code Editor
- Sample Files and Posts
- 700 Tips & Best Practices!
- And much more...

MecSoft Corpotation

Table of Contents

Read This First!

How to use this Guide:

1. Newer videos are always listed higher in the tables and in the **Table of Contents** on the left side of this guide.
2. Older videos that are listed closer to the middle and bottom are still relevant since all features introduced are still in the product today.
3. Select a link from column #1 to go to the page in this guide dedicated to that video. Each video page contains a video description, video time-stamp links into the actual video, a list of **Tips & Best Practices** and **Key Takeaways** extracted from each video.
4. If you are looking for a specific feature, see the Product Features page. It lists many of the features in each product with a link to the What's New webinar that first introduced the feature.
5. There are over **700 Tips & Best Practices** in this entire guide! See each video page for the list of Tips & Best Practices demonstrated.

About the Table Below:

1. The table below lists the full-length training webinar in the left side column #1. The video titles shown are abbreviated and are not the full titles shown on the MecSoft YouTube channel.
2. The icons in columns #2-#6 indicate the MecSoft CAM configuration (XPRESS, STD, XPERT, PRO, & PREM) that is relevant to each video.
3. It is important to note that ALL of the CAM features demonstrated in these videos are supported in both RhinoCAM and VisualCAD/CAM and are identical in their implementation and work flow.
4. The icons in columns #7 and #8 simply indicate which product (RhinoCAM or VisualCAD/CAM) is demonstrated in the video.
5. If there is an icon in Column #9, it means that the video demonstrates VisualCAD functions during the video.

Table of Webinars by Configuration and Product:

#1 Webinar	#2 XPRESS	#3 STD	#4 XPERT	#5 PRO	#6 PREM	#7 Rhino	#8 VCC	#9 Visual CAD
Configurations	-	▶	▶	▶	▶	▶	-	-
2.5 Axis Support	▶	▶	▶	▶	▶	▶	-	-
4 Axis Continuous	-	-	▶	▶	▶	▶	-	-

#1 Webinar	#2 XPRESS	#3 STD	#4 XPERT	#5 PRO	#6 PREM	#7 Rhino	#8 VCC	#9 Visual CAD
4 Axis Indexed	-	-	▶	▶	▶	▶	-	-
3 Axis Basic	▶	▶	▶	▶	▶	▶	-	-
2.5 Axis Adv	▶	▶	▶	▶	▶	▶	-	-
TURN Advanced	-	▶	▶	▶	▶	-	▶	▶
2.5 Axis 2D Dwgs	▶	▶	▶	▶	▶	-	▶	▶
NEST Real World	-	▶	▶	▶	▶	▶	-	-
VisualCAD/CAM	▶	▶	▶	-	-	-	-	-
Getting Started	▶	▶	▶	▶	▶	▶	-	-
5 Axis Indexed	▶	▶	▶	▶	▶	▶	-	-
Cabinet Automation	-	▶	▶	▶	▶	▶	-	-
Using VisualCAD	-	-	-	-	-	-	-	▶
Repair Control Geometry	-	-	-	-	-	-	-	▶
CAM a 2D Drawing	▶	▶	-	-	-	-	-	▶
Guitar Body	-	-	-	▶	▶	▶	-	-
Real World Parts	-	-	-	▶	▶	▶	-	-
Power User	▶	▶	▶	▶	▶	▶		
4 Axis Advanced	-	-	▶	▶	▶	▶	-	-
3 Axis Advanced		▶	▶	▶	▶	▶		
3 Axis Basic	-	▶	▶	▶	▶	▶	-	-
2.5 Axis Prismatic	-	▶	▶	▶	▶	▶	-	-
2.5 Axis Support	▶	▶	▶	▶	▶	▶	-	-

#1 Webinar	#2 XPRESS	#3 STD	#4 XPERT	#5 PRO	#6 PREM	#7 Rhino	#8 VCC	#9 Visual CAD
Cut Tubular Intersections	-	-	-	-	-	-	-	
CAM Automation								
TURN Basic	-						-	-
Flip Machining	-						-	-
Post Advanced							-	-
Profile NEST	-						-	-
Model Masking Surfaces	-	-	-	-	-	-	-	
2.5 Axis Bootcamp	-						-	-
2 & 3 Axis	-						-	-
2.5 Axis Special	-	-	-				-	-
Performance								
Molds	-	-	-				-	-
Guitars	-	-	-				-	-
Control Geometry	-						-	-
5 Axis Indexed	-	-	-				-	-
4 Axis Introduction	-	-					-	-
4 Axis Advanced	-	-					-	-
4 Axis Basic								
3 Axis Power	-						-	-
3 Axis Advanced	-	-	-				-	
VisualCAD Shorts	-	-	-	-	-	-	-	
3 Axis Basic							-	-

#1 Webinar	#2 XPRESS	#3 STD	#4 XPERT	#5 PRO	#6 PREM	#7 Rhino	#8 VCC	#9 Visual CAD
CAM in Edu						-		
2.5 Axis Power User	-						-	-
2.5 Axis Advanced	-						-	-
Scanned Orthotics								
Mesh, CAM & ART	-					-		
4 Axis Scan Data								
CAM in High School						-		
G Code Editor	-						-	-
Profile-NEST	-						-	-
2.5 Axis Pocketing & Profiling							-	
Common Support							-	-
RhinoCAM in Edu							-	-
Knowledge Bases	-						-	-
Using VisualCAD	-	-	-	-	-	-	-	
Hole Machining	-						-	-
Post Advanced	-						-	-
TURN Basic	-						-	-
Use of Selections	-					-		
5 Axis Machining	-	-	-	-			-	-
Multi Sided Parts	-	-	-					
Parts Nesting	-						-	-

#1 Webinar	#2 XPRESS	#3 STD	#4 XPERT	#5 PRO	#6 PREM	#7 Rhino	#8 VCC	#9 Visual CAD
Machining Molds	-	-	-				-	-
Mesh Machining	-						-	-
Feature Machining	-					-		
Cutting Tools								
3 Axis Best Practices	-							
CAM in Education	-						-	-
Feeds and Speeds								
Coordinate Systems								
2.5 Axis Standard	-					-		
4 Axis Jewelry	-	-					-	-
CAM in Edu	-						-	-
CAM Automation	-						-	-
Feature Machining	-							
4 Axis Furniture	-	-				-		
CAM in Education	-						-	-

Reference Material

MecSoft places a wide range of reference material at your fingertips! You can choose from MecSoft Case Studies, Tech Blogs, Play Lists, Quick Start Guides, Online Web Help, Resource Hub, Community Forums and VisualServe Portal! Make a selection below.

MecSoft Reference Materials:

[MecSoft Resource Hub]

[MecSoft Play Lists]

[MecSoft Tech Blogs]

[MecSoft Case Studies]

[MecSoft Forums]

[MecSoft VisualServe]

Product Features

Below is a list of product features that have been implemented in both RhinoCAM and VisualCAD/CAM. They are listed in the order of their introduction with newer features being listed first. The features below are in addition to the base machining operations and simulation features inherent in the product. The links take you to the What's New webinar where the feature is discussed in detail when introduced.

1. **3+2 Auto Roughing is a Game-Changer** — The new operation automatically analyzes the stock and part, determines the optimal indexing angles, creates multiple setups, and generates adaptive roughing toolpaths — all with almost zero manual input. Ideal for complex prismatic or semi-prismatic parts.

2. **4-Axis Extrusion Methods Open New Possibilities** — Extrusion Machining (Profile + Rail) is perfect for parts with constant cross-section (no need for a full 3D model). Extrusion Projection Machining solves the long-standing limitation of features that extend above or below the rotary axis by projecting the extruded profile onto the actual 3D geometry.

3. **2.5-Axis Auto Deburring Works with Ball Mills** — You no longer need special V-mills or chamfer tools. The new operation automatically detects edges and generates safe Deburring/chamfering paths with full gouge checking.

4. **Tri-Dexel Simulation is a Major Performance Win** — If you have an NVIDIA GPU (especially in Professional+ configurations), enable the new Tri-Dexel simulation model. It delivers 5–10× faster simulation with excellent accuracy — especially noticeable on complex multi-setup jobs.

5. **Per-Setup Fixtures & Clearances** — You can now assign different fixtures and clearance distances to individual setups instead of applying one setting to the entire job. This is extremely useful for tombstone or multi-part fixtures.

6. **Arc Feed Rate Optimization Improves Finish & Tool Life** — Reduce feed rates automatically before and after arcs (corners). This is configurable by arc length and radius and helps prevent chatter and extend tool life on high-speed finishing passes.

7. **Drill Point Sorting Reduces Non-Cutting Time** — The new “Drill Point Sorting Between Operations” option makes the next operation start at the closest point to where the previous operation finished — a simple but effective way to shorten cycle time on parts with many holes or features.

8. **Python Post-Processors Are Now Extremely Powerful** — 2026 significantly expands Python post support, including full drill bank control, dual-head machines, and non-standard G-code formats (Holzher, WoodFlash, Flow Master waterjet, etc.). Custom post development is now much more accessible.

9. **Expanded API for Automation** — New API functions allow deeper integration with external scripts for setups, operations, simulation, and Knowledge Base workflows — great for shops building custom automation pipelines.

10. **Cutter Compensation Now Works in Hole Pocketing** — You can now use G41/G42 in hole pocketing and profiling operations for tighter control and better finish on production holes.
11. **Thread Milling Got Smarter** — New controls for **Start Depth (Relief Depth)**, pitch (per mm or per inch), and **Number of Starts** give you much more precise control over thread creation.
12. **Feedrate Optimization is Now Per-Operation** — Corner feed reduction settings have moved directly into the operation dialog (instead of global preferences), making it easier to fine-tune per toolpath.
13. **Faceting Controls Improve Toolpath Quality** — New mesh aspect ratio and minimum edge length settings in 3-axis operations help prevent jagged or erratic toolpaths on complex ruled surfaces.
14. **Drive Curves + Drive Surfaces Can Now Be Used Together** — In operations like Parallel Finishing, you can select both surfaces and curves as drive geometry — very useful for machining between nested regions.
15. **Corner Cleanup Loops in Projection Pocketing** — New option (Professional configuration) adds automatic cleanup loops at sharp corners to reduce material retention and machine jerking.
16. **Clear Flats Now Supports Stock Allowance** — You can now leave a specified stock allowance on flat areas when using the “Clear Flats” option, allowing for a dedicated finishing pass.
17. **Adaptive Roughing Improvements** — Helical entries at every cut level and arc motions for transfers/exits make Adaptive Roughing more efficient and tool-friendly.
18. **Rigid Tapping (G84) is Now Supported** — Higher-accuracy tapping with synchronized spindle and feed control.
19. **Machine Tool Linear Limits Detection** — Define X/Y/Z travel limits in the Machine Tool setup. The software now visually shows the limits and flags any violations as errors during simulation.
20. **Copy/Clone Naming is Now Customizable** — You can disable or change the automatic “-copy” and numbering behavior when cloning operations.
21. **Maintain Order of Control Geometry** — New option prevents the software from automatically merging or reordering selected curves/regions — useful when order matters.
22. **Python 3 SDK Support** — Major upgrade from Python 2, plus non-interactive execution for “lights-out” automation.

23. **Post by Tool Number (Ascending/Descending)** — Enable this in CAM Preferences > Post-Processing to automatically reorder operations by tool number when posting. This dramatically reduces tool changes on the machine without manually reorganizing your Machining Job tree.

24. **Always Save & Load Post with the File** — Check both “Always Save Post to File” and “Always Load Post from File” in CAM Preferences. This locks the correct post-processor to each job file, preventing errors when moving files between computers or after software updates.

25. **Tool Contact Condition is a Major Win for Mold Makers** — The new “Contact” surface condition makes the tool stop exactly when the ball touches the perimeter edge (parting line). This prevents the tool from riding up over the parting line (which causes flash) and often eliminates the need for masking surfaces.

26. **Skim Clearance Now Considers In-Process Stock** — Retracts are now calculated based on remaining stock, allowing much lower (and faster) retract heights while still avoiding collisions.

27. **Negative Offsets in Face Off** — You can now enter negative values for cut area extension. This tightens the toolpath around the part edge, saving significant machining time on facing operations.

28. **High-Speed Arc Entries/Exits Now Work in Facing** — Facing operations now benefit from the same smooth 2D/3D arc entries, exits, and transfers previously available only in pocketing — reducing machine jerk and improving finish.

29. **Exact Z-Level Distances** — You can now specify precise cut distances for Z-level operations instead of being forced to cut at the absolute bottom level. This gives much more control over step-downs.

30. **True 5-Axis Drilling** — The tool can now stay normal to the surface during drilling on curved or angled geometry (not just indexed 5-axis drilling).

31. **4-Axis Rotation on Z-Axis** — You can now assign the 4th axis to rotate around the positive or negative Z-axis (C-axis style), expanding 4-axis capabilities for certain machine configurations.

32. **Recalculate Angles After Reordering** — After moving operations within a setup, use the new Recalculate Angles option to update machine tool kinematics without regenerating the entire simulation.

33. **Improved Network Lock License Manager** — The new interface automatically creates firewall rules and generates registry files for client machines, making network license setup much easier.

34. **Drilling Along Curves is a Huge Time Saver** — You can now select any 2D or 3D curve (including surface edges and circles) and the software will automatically generate drill points along it. No more manually creating point arrays.

35. **Clustering in Profiling Prevents Tool Interference** — When profiling nested parts, the new **Clustering** option alternates inside/outside cutting to keep the tool from colliding with adjacent parts.
36. **Gouge Checking for Profiling** — A powerful new option that automatically calculates the safe outer perimeter of selected curves so the tool never gouges into neighboring geometry.
37. **Projected Spiral Offset Pocketing (Professional+)** — New cut pattern that combines spiral entry with offset pocketing for superior finish on complex pockets.
38. **Improved Horizontal Roughing ("Clear Flats")** — The software now ensures safe entry motions outside of material when clearing flat areas, preventing plunging into uncut stock.
39. **4-Axis Drilling Along Curves** — You can now drill along curves on 4-axis parts without manually creating indexed setups — the software handles the rotary motion automatically.
40. **5-Axis Alternate Rotation Angles** — The post can now output both the primary and alternate (180° offset) solutions for rotary axes, giving you more flexibility on machines with limited rotation range.
41. **Turning: Geometry No Longer Needs to Touch X-Axis** — You can now machine parts that are offset from the X-axis (great for parting-off operations behind the part).
42. **Linear Retracts for ID Machining** — ID turning now uses linear retracts to the clearance plane instead of diagonal moves, reducing the chance of tool interference.
43. **Save Post with File** — Enable **"Always Save Post to File"** and **"Always Load Post from File"** in preferences. This locks the correct post-processor to each job file and prevents errors when moving files between computers.
44. **New License Service Manager** — The 2023 licensing system includes a much-improved Network Log License Manager with better diagnostics and automatic firewall rule creation.
45. **G-Code Editor is Now Fully Integrated (AMS Subscribers)** — The G-Code Editor is no longer a separate module. It is built directly into the Milling module. You can now automatically back-plot and simulate the actual posted G-code in real time on your part — a massive improvement for catching post-processor errors before running on the machine.
46. **Programmable Post Processors (Python)** — 2022 introduced full Python-based programmable posts. This gives advanced users and post developers far more flexibility to create custom output for complex machines (drill banks, dual heads, special formats, etc.).
47. **New Thread Milling Operation** — A dedicated Thread Milling operation was added, giving much better control over thread creation compared to previous workarounds.

48. **New Saw Cutting Operation** — A dedicated operation for saw blades (disk cutting) with proper tool definition and containment options — very useful for panel saws and certain router applications.
49. **Hole Pocketing Improvements** — Better handling of large holes with improved helical entry and cleanup pass options.
50. **Save/Load Post with File** — A major usability win: You can now save the post-processor settings with the part file so the correct post is automatically loaded when the file is opened on any computer.
51. **Knowledge Base Enhancements** — Improved saving and loading of Knowledge Bases with better support for complex multi-operation workflows.
52. **Re sizable Dialogs** — Almost all dialogs became re sizable, significantly improving usability on high-resolution or multi-monitor setups.
53. **Re sizable Dialogs** — Almost every dialog in the software became re sizable. This was one of the most requested usability improvements and significantly improved the experience on high-resolution monitors.
54. **New Saw Cutting Operation** — A dedicated operation was added for saw blades (disk cutting). This made it much easier to program panel saws and certain router applications with proper tool definition and containment.
55. **Improved Horizontal Roughing** — Better handling of flat areas and improved entry/exit motions, especially when using the “Clear Flats” option.
56. **Knowledge Base Improvements** — Enhanced saving and loading of Knowledge Bases, making it easier to create and reuse automated workflows.
57. **Post-Processor Enhancements** — New macros were added, giving users more control over G-code output, especially for multi-axis and custom machines.
58. **Performance Optimizations** — Toolpath calculation and simulation were noticeably faster, especially on complex parts with many operations.
59. **4-Axis & 5-Axis Improvements** — Several refinements to indexed and continuous multi-axis toolpaths, including better handling of tool orientation and collision avoidance.
60. **Save Post with File Option** — Users could now save the post-processor settings with the part file (this became even more refined in later releases).
61. **G-Code Editor Module is a Major Leap Forward** — For the first time, AMS subscribers could load the actual posted G-code and simulate it directly on the part geometry with full cut material simulation. This was a game-changer for catching post-processor errors before sending code to the machine.
62. **Programmable Post Processors (Python)** — 2020 introduced the Python-based programmable post processor framework. This gave advanced users and post developers

unprecedented flexibility to create custom output for complex machines (drill banks, dual-head machines, special formats, etc.).

63. **Knowledge Base Improvements** — Significant enhancements were made to how Knowledge Bases are saved and loaded, including better support for complex multi-operation workflows and improved automation capabilities.
64. **Re sizable Dialogs** — Almost every dialog in the software became re sizable. This was one of the most requested usability improvements and made working on high-resolution monitors much more comfortable.
65. **Save Post with File Option** — Users could now save the post-processor settings with the part file, ensuring the correct post is automatically loaded when the file is opened on any computer.
66. **Performance Improvements** — Toolpath calculation and simulation were noticeably faster, especially on complex jobs with many operations.
67. **New Licensing System** — The 2020 release introduced improvements to the licensing infrastructure (Network Lock enhancements and better diagnostics).
68. **Feature-Based Machining + Silhouette Features** — One of the biggest productivity wins. The software can now automatically detect pockets, slots, holes, chamfers, **and silhouettes**, then generate complete toolpaths with a single command.
69. **Drag Knife Operation is Excellent for Sign Making** — New dedicated operation with automatic swivel control and corner limit angle settings — perfect for vinyl, fabric, and thin material cutting.
70. **Cornering Actions Give Precise Control** — New options (Dog bone, T-bone, Loop, Sharp Limited) allow you to control exactly how the tool behaves at internal and external corners during profiling.
71. **Clustering (Inside-Out Cutting)** — Prevents parts from separating too early during nested profiling by cutting from the inside out. Very useful for keeping small parts attached to the sheet until the very end.
72. **Automatic Stock Extent in Facing** — Facing operations now automatically use the stock silhouette, eliminating the need to manually select containment regions in most cases.
73. **Trichoidal (High-Speed) Pocketing** — New cut pattern option for smoother, faster, and more tool-friendly pocketing, especially in harder materials.
74. **Shank/Holder Collision Detection** — Simulation now clearly highlights collisions with the tool shank and holder in red and marks the operation as “dirty” so you know it needs review.
75. **Clear Flats with Fillets/Chamfers** — The “Clear Flats” option now properly handles parts with fillets and chamfers without gouging.

76. **Faster Pencil Tracing** — Up to 4–5× faster computation with a new “minimum pet length” parameter.
77. **Skin Clearance Now Supports 3D Geometry** — Retract heights are calculated more intelligently when 3D features are present, reducing unnecessary Z moves.
78. **Nesting Solid Models + Explode Cabinet Utility** — You can now nest actual 3D solids, surfaces, and meshes. The new “Explode Cabinet Design” utility automatically flattens 3D cabinet assemblies into 2D panels ready for nesting — a huge time saver for cabinet shops.
79. **Save Post with File** — A major usability improvement that lets you lock the correct post-processor to each job file.
80. **Automatic Feature Detection & Machining (AFD/AFM) is the Star Feature** — This was the biggest addition in 2018. After creating a Knowledge Base, you can detect all machinable features on a solid model (in all orientations) and generate complete toolpaths with one click. This dramatically reduces programming time on prismatic parts.
81. **Interactive Feature Machining** — Even if you don’t use full automation, you can right-click any detected feature and instantly choose the operation type (pocketing, profiling, drilling, etc.) with parameters already populated.
82. **ART and NEST Modules Are Now Free** — One of the most impactful changes: the ART (artistic relief) and NEST (nesting) modules became included at no extra cost with all Mill configurations starting in 2018.
83. **Ghosted Display Mode** — A very useful new display option that makes hidden features (like internal cavities) visible without having to hide the outer geometry.
84. **Chain Selection (Shift + Click)** — You can now quickly select chains of curves or surface edges by holding Shift — a major time saver when selecting containment or profiling geometry.
85. **Holder Collision Detection** — Simulation now detects and highlights collisions with the tool shank and holder (shown in red) and marks the operation as “dirty” so you know it needs attention.
86. **“Create Round” 4-Axis Operation** — A simple but powerful new operation that automatically turns rectangular stock into a cylinder using the 4th axis — very useful for preparing stock for round parts.
87. **Follow Containment in Horizontal Finishing** — Prevents unnecessary retracts by making the tool follow the containment boundary instead of retracting and re-engaging.
88. **Peck Tapping Operation** — New dedicated operation that allows you to specify peck depth for tapping deep holes.
89. **Mesh Module Introduced** — A completely new module for working with meshes (point cloud conversion, Boolean operations, decimation, smoothing, and automatic hole repair) — useful for reverse engineering and 3D printing workflows.

90. **RhinoCAM API** — The introduction of a Python/.NET API opened the door for custom automation and integration with other systems.
91. **Automatic Feature Detection (AFD) was the biggest new feature** — This was the very first version of what later became Automatic Feature Machining (AFM). It could automatically detect pockets, holes, slots, chamfers, and planar faces on solid models — a major step toward reducing manual geometry selection.
92. **Knowledge Bases + Feature Detection = Powerful Automation** — Even in its first version, combining Knowledge Bases with Automatic Feature Detection allowed users to program many prismatic parts with significantly less manual work.
93. **Swarf Machining Introduced** — The new Swarf operation (keeping the side of the tool in contact with a ruled surface) was a major addition for 5-axis users, especially in mold and aerospace work.
94. **Flow Curve Operation** — A new 5-axis strategy that allows the tool to follow a user-defined curve while staying normal to the surface — very useful for complex organic shapes.
95. **Improved Knowledge Base Management** — Better tools for saving, organizing, and loading Knowledge Bases made automation more practical for daily use.
96. **Performance Improvements** — Toolpath calculation and simulation were noticeably faster, especially on complex 3-axis and 5-axis jobs.
97. **Usability Enhancements** — Several small but meaningful improvements were made to the interface and workflow (better icons, display options, and dialog behavior).
98. **Turning Module Refinements** — Several behind-the-scenes improvements were made to the TURN module for better reliability and output quality.

AMS Webinars RC & VCC

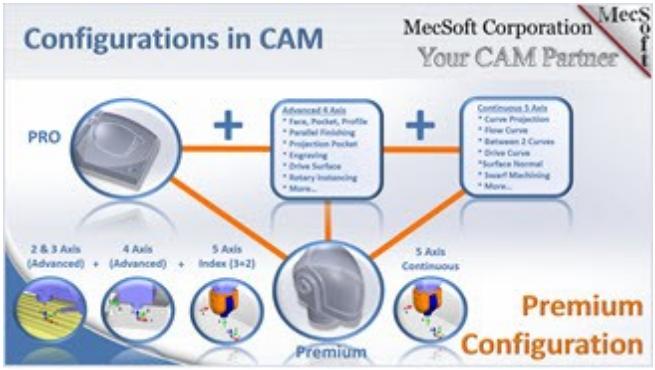
In this section you will find a complete list of full-length AMS Training Webinars. Newest webinars are listed first.

4.1 Configurations

Link: https://www.youtube.com/watch?v=PEY3iIsIcy8

This is one of the most important webinars for anyone considering or currently using MecSoft CAM. It clearly explains the five product configurations (Express → Standard → Expert → Professional → Premium) and the available modules (Mill, Turn, Nest, ART, etc.), helping users choose the right level for their needs and budget.

Source Files: 20.24.10 AMS Webinar: Configurations & Modules in CAM.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Product Overview	RhinoCAM vs VisualCAD/CAM differences and target users
6:00	Demo Mode & Testing Configurations	How to test any configuration without a license
10:00	Express Configuration (2.5 & Basic 3-Axis)	Features, limitations, and ideal use cases
13:00	Standard Configuration (Enhanced 2 & 3-Axis)	Added operations, better finishing, tapping/boring
19:00	Expert Configuration (4-Axis)	Continuous & indexed 4-axis, drive surface machining

Time	Topic / Section	Description
24:00	Professional Configuration (Advanced 3-Axis)	21 advanced 3-axis operations, toolpath editing, collision detection
34:00	Premium Configuration (5-Axis Continuous)	Full 5-axis operations (Swarf, surface normal, flow curve, etc.)
40:00	Modules Overview (Turn, Nest, ART, AMS Exclusives)	What each module adds and AMS-only features
48:00	Live Demos of Each Configuration	Real part examples for Express through Premium
1:01:00	Upgrade Path & Best Practices	How to upgrade + tips for choosing the right level
1:03:00	Conclusion & Resources	Next steps, support, and upcoming webinars

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Test Before You Buy** — Disconnect from the internet, release your license (if cloud-based), and run any configuration in Demo Mode to explore features before purchasing.
2. **Upgrade Path is Easy & Affordable** — You can upgrade from any lower configuration to a higher one by paying only the difference plus a small premium. No need to repurchase the entire product.
3. **Choose Based on Your Parts, Not Just Budget**:
    - Express/Standard → Prismatic parts, sheet work, basic contours
    - Expert → Parts that benefit from a 4th axis (rotary features)
    - Professional → Complex molds, high-detail contours, advanced toolpath control
    - Premium → True 5-axis continuous work (aerospace, molds, complex surfaces)
4. **RhinoCAM vs VisualCAD/CAM**:
    - Choose RhinoCAM if you need powerful CAD modeling (Grasshopper, SubD, API, advanced translators).

- Choose VisualCAD/CAM if you mainly import files from other CAD systems and want simpler, lower-cost CAD.

5. **AMS Subscribers Get Extras** — Profile Nesting and the full G-Code Editor (edit any G-code, not just MecSoft output) are exclusive to active AMS customers.
6. **Use Opposite-Direction Finishing** — For circular or radial features, run spiral and radial finishing in opposite directions for significantly better surface finish.
7. **Knowledge Bases Save Time** — Save your favorite operation setups as Knowledge Bases so you can reuse them instantly on future jobs.
8. **Simulation is Visual Only** — The high-end polygonal simulation looks great but is for visual verification only — always verify critical dimensions on the machine.
9. **Mesh Module is Discontinued** — Do not rely on the old Mesh module for point cloud work.
10. **Start Simple, Upgrade Later** — Many users begin with Standard or Expert and upgrade as their work becomes more complex or they add rotary axes.

Key Takeaways

This webinar removes the confusion around MecSoft's product lineup. The configurations are tiered and logical — each level adds meaningful capabilities without forcing you to buy features you don't need.

Best advice from the session:

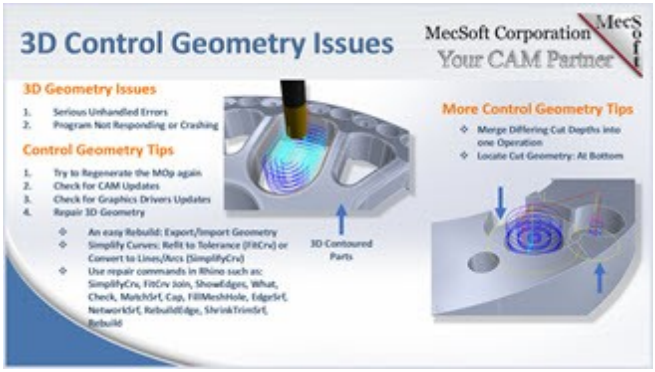
- Understand the differences between configurations before purchasing.
- Select the simplest configuration that meets your machining needs.
- Learn shared CAM features before upgrading to advanced modules.
- Match configuration capabilities to real-world production requirements.
- Upgrade modules gradually as workflows become more advanced.
- Use standardized workflows to improve programming efficiency.
- Apply recommendations from the live Q&A session when evaluating CAM solutions.

4.2 2.5 Axis Support

Link: https://youtu.be/GO60xgjbILg

This is a **support-focused training webinar** that addresses the most common issues users encounter when programming 2.5-axis jobs in RhinoCAM and VisualCAD/CAM. It covers machining problems, post-processing errors, geometry/setup issues, CAM preferences, performance problems, and practical troubleshooting steps.

Source Files: 20.26.04 AMS Webinar 2 5 Axis Support Issues.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of common 2.5-axis support issues
5:00	Machining Issues	Gouging, incorrect depths, poor surface finish, machine errors
12:00	Post-Processing Issues	Missing posts, wrong extensions, no G-code output
18:00	Geometry Setup Issues	Part orientation, stock definition, Work Zero problems
24:00	CAM Preference Issues	Tool libraries, machining preferences, simulation settings
32:00	System & Hardware Performance	CPU, GPU, RAM, Windows settings, driver recommendations
40:00	Toolpath Generation & Simulation Performance	Tolerance impact, geometry complexity, TriDexel vs Voxel
48:00	2.5-Axis Specific Issues	Open loops, cut sides, arc fitting, drilling cycles
55:00	Troubleshooting Error Messages	Common errors and how to fix them

Time	Topic / Section	Description
1:05:00	Support Resources & Best Practices	How to contact support effectively + final tips

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Always document the actual G-code file** that caused the machine error (not just the internal toolpath).
2. **Use third-party G-code simulators** to verify posted code before running on the machine.
3. **Check the basics first**: Part orientation, stock definition, Work Zero, and stock visibility.
4. **For poor surface finish**: Loosen tolerances and step-overs during roughing; tighten only for finishing.
5. **Fix "Open Loops" errors**: Merge all control curves into single closed loops.
6. **Fix "Invalid Data" errors**: Ensure stock is defined and tool diameters match between roughing and re-roughing operations.
7. **Save preferences to a file** instead of the registry when working with multiple machines.
8. **Enable "Always post-process during toolpath generation"** to quickly view G-code via the G-code icon.
9. **Use Voxel simulation mode** for fast roughing previews; switch to TriDexel for final verification (requires NVIDIA GPU).
10. **Prioritize CPU clock speed** over core count for faster toolpath calculations.
11. **Reboot daily** and run Rhino/CAM in "Real Time" priority in Windows Task Manager.
12. **For drilling issues**: Check the option "Always output drill cycles as linear motions" if your machine doesn't support canned cycles.
13. **When contacting support**: Attach the part file, isolate the problematic operation, include the actual G-code file, and describe the exact error message.

Key Takeaways

This webinar is an excellent **troubleshooting guide** for 2.5-axis users. Most support issues fall into a few categories:

- Geometry/setup problems (especially with 2D drawings)
- Post-processor and preference configuration

- Performance bottlenecks (hardware + settings)
- Common error messages that have simple fixes

The biggest message: **Most problems are preventable** with proper setup, simulation, and documentation.

4.3 4 Axis Continuous

Link: https://www.youtube.com/watch?v=mDf7TAzyMOA

This is a full-length training webinar focused on Continuous 4-Axis machining of 3D models in RhinoCAM / VisualCAD/CAM. It uses a real-world ring part with diamond patterns as the example and covers everything from machine setup to G-code output.

Source Files: 20.25.11 AMS Webinar Continuous 4 Axis of 3D Models.zip

Topic Index with Timestamp Links

Times below are **approximate** based on the webinar's natural flow and transcript structure. Click any timestamps to jump directly to that section.

Time	Topic / Section	What's Covered
0:00	Introduction & Agenda	Welcome by Joe, overview of the session, goals for continuous 4-axis machining
10:00	Geometry Requirements & Machine Setup	Supported geometry (solids/surfaces/meshes/curves), 4-axis machine definition, rotary axis on table, origin setup
15:00	R-Level Roughing (Demo starts)	Radial-level roughing strategy, step-down, stock to leave, simulation

Time	Topic / Section	What's Covered
25:00	Parallel Finishing for Roughing	Using parallel finishing as a roughing pass, single entry/exit, uniform radial steps
35:00	R-Level Finishing	Finishing in radial levels on protruding diamond features, tight step-down (0.05 mm)
45:00	Parallel Finishing (Finishing pass)	Cross-cut finishing pattern, containment adjustments, final surface quality
55:00	4-Axis Pocketing	Pocketing diamond areas with flat-bottom/vertical walls, negative stock to separate features, cleanup pass
1:05:00	Facing & Profiling	Facing between curves, ramp entry, profiling/offset cleanup on edges
1:20:00	Drive Surface Machining (True 4-Axis)	Most important advanced section — tool stays normal to surface on non-centered/overhanging geometry
1:40:00	Post-Processing & G-Code Review	Laguna HHC post-processor, A-axis output, coordinated X/Y/Z/A moves, G1 rotary commands
1:50:00	Conclusion & Q&A Wrap-up	Final remarks and holiday wishes

Key Tips & Best Practices

Here are the most valuable practical tips shared during the webinar:

1. **Avoid containment geometry** in 4-axis operations when possible — it slows calculations dramatically. Instead, adjust your stock size to control the machining area.
2. **Use Parallel Finishing for roughing** — it creates a single entry/exit and reduces unnecessary retracts and transfers compared to traditional level roughing.
3. **Stock settings matter**:
    - Set stock to **zero** to keep connected patterns intact.
    - Use **negative stock** (e.g., -0.07 mm or -0.1 mm) in pocketing to cleanly separate diamond features.

4. **Drive Surface Machining** is the key to "true" 4-axis work — it keeps the tool normal (perpendicular) to the surface even on parts that extend beyond the radial axis. Essential for non-centered or complex topologies.
5. **Tool choice**:
    - Stick with **ball mills** or **flat end mills** for most 4-axis work.
    - Avoid tapered tools unless necessary — they significantly increase calculation time.
6. **4-axis pocketing requirements**: The pockets must have flat bottoms and vertical walls for reliable results.
7. **Simulation best practices**: Use tight tolerance and fine accuracy settings, especially on small parts. Color-code the stock for clear material removal visualization.
8. **Use curves** for pocketing and profiling boundaries (same as 3-axis workflows).
9. **R-Level operations** excel at machining protruding features level-by-level from the outside in.
10. **Post-processor requirement**: Your post must fully support 4-axis output (A-axis rotary moves). The webinar demonstrates clean G-code with coordinated X/Y/Z/A motion.
11. **Machine setup tip**: Always set the rotary center at the world origin (0,0,0) and confirm positive rotation direction matches your physical machine.

Key Takeaways

This webinar is one of the best practical guides available for **Continuous 4-Axis machining** in MecSoft CAM products. It clearly shows when to use each strategy:

- Align the model correctly with the rotary axis.
- Verify tool orientation before generating toolpaths.
- Optimize stepover and tolerance settings.
- R-Level Roughing/Finishing → Best for radial, protruding features
- Parallel Finishing → Excellent for uniform coverage and roughing with minimal retracts
- Drive Surface → The "secret weapon" for true surface-normal 4-axis machining on complex or offset parts
- Use simulation to validate rotary motion.
- Confirm post processor output before machining.
- The session moves from basic concepts all the way to verified G-code output, making it extremely useful for both new and experienced users.

4.4 4 Axis Indexed

Link: https://www.youtube.com/watch?v=f04bEQMWAAo

This is the companion webinar to the November “Continuous 4 Axis” session. It focuses on Indexed 4-Axis machining — rotating the table to fixed angles (0°, 90°, 180°, 270°), then performing standard 2.5-axis or 3-axis operations in each position. The example part is a U-shaped table leg machined from square stock with dowel holes, decorative scallops, curved profiles, and tapered walls.

Source Files: 20.25.10 AMS Webinar Indexed 4Axis of 3D Models.zip

Topic Index with Timestamp Links

Times are **approximate** based on the natural webinar flow and agenda order. Click any link to jump to that section of the video.

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of indexed vs continuous 4-axis, part example (U-shaped table leg), goals
8:00	Geometry Requirements	What geometry is needed (2D curves for 2.5-axis, 3D model for 3-axis), adding protective cylinders
12:00	How Indexed 4-Axis Machining Works	“Rotate Table Setup” concept — index to fixed angles then machine normally
18:00	Recommended Cutting Tools	3/4" flat end mill (rough/finish/drill), 2" ball mill (engraving scallops)
22:00	4-Axis Machine Definition & Setup	Table-type 4th axis, A-axis alignment, stock definition (square bar + oversize)

Time	Topic / Section	Description
28:00	Rotate Table Setups (0°, 90°, 180°, 270°)	Creating indexed setups in RhinoCAM
35:00	Roughing & Finishing Demos (Multiple Orientations)	3-axis Horizontal Roughing + Finishing in each indexed position
50:00	Drilling Operations	Using end mill as drill on cylinder features, flat-bottom holes
58:00	2-Axis Engraving (Scallops)	Ball mill tip-only engraving on internal curves
1:05:00	Cut Material Simulation	Full multi-setup simulation, collision checking
1:12:00	Post-Processing & G-Code	WinCNC post, A-axis output (A0, A90, A180, A270), grouping by tool
1:20:00	Conclusion & Best Practices Recap	Final tips and workflow summary

Key Tips & Best Practices

Here are the most useful practical takeaways from the webinar:

1. **Cylinders are your friends** — Add temporary cylinders to the ends of the part/stock to prevent 3-axis tools from over-cutting into areas you want to protect.
2. **Indexed 4-axis is simpler than continuous** — You only need standard 2.5-axis or 3-axis toolpaths after each table rotation. No complex multi-axis toolpaths required.
3. **Geometry rules**:
    - 2.5-axis operations → only need 2D curves
    - 3-axis operations → need the 3D model
4. **Stock setup tip**: Make stock slightly larger than the final part (+1/4" on width/height) and align it to the part center.
5. **Rotate Table Setup** is the key command — create one setup per desired angle (0°, 90°, 180°, 270°). The table stays fixed during each operation.
6. **Tool strategy**:
    - One 3/4" flat end mill can handle roughing, finishing, **and** drilling.

- Use a large ball mill (2") for decorative engraving/scallops with tip-only cutting (cut depth = 0).

7. **Step minimization** (set to ~10%) dramatically improves roughing efficiency.
8. **Group operations by tool** in the Machining Job tree — especially useful if you have an Automatic Tool Changer (ATC).
9. **Post-processor must support multi-axis** — it needs to output the A-axis rotation commands (A0, A90, etc.) before each setup.
10. **Always run full simulation** across all indexed setups before posting G-code to catch any collisions or missed material.
11. **Final parts** are usually cut off manually from the remaining stock after the 4-axis operations are complete.

Key Takeaways

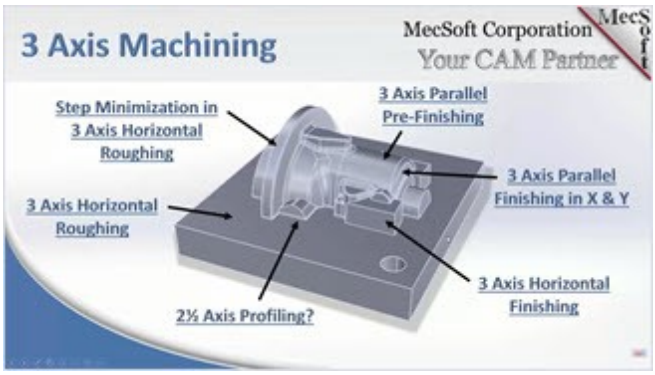
- This webinar is the perfect “Indexed 4-Axis 101” companion to the Continuous 4-Axis session. It shows how you can achieve complex multi-sided machining on a 4-axis machine without needing continuous rotary toolpaths — just smart use of Rotate Table Setups + standard 2.5/3-axis strategies.
- Best for: Parts with features on multiple faces (top, bottom, sides) that can be machined at 90° increments.
- Use indexed 4-axis machining when simultaneous rotary motion is not required.
- Confirm rotary index angles and work offsets before generating toolpaths.
- Reuse proven 3-axis machining strategies at indexed positions.
- Validate each setup with simulation.
- Test post processor output and machine commands before production.

4.5 3 Axis Basic

Link: https://www.youtube.com/watch?v=f-X6ask5p-A

This webinar focuses on practical 3-axis machining of complex 3D models (specifically a mold core insert) using MecSoft CAM Standard (RhinoCAM / VisualCAD/CAM). It emphasizes efficient roughing, high-quality finishing, and proper handling of critical features like parting planes.

Source Files: 20.25.09 AMS Webinar Machining 3D Models with 3 Axis Standard.zip



Topic Index with Timestamp Links

Times are approximate based on the webinar's content flow and agenda. Click any time stamp to jump directly to that section.

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Welcome, agenda overview, part introduction (mold core insert)
3:15	Geometry Requirements	Analyzing parting plane, fillets, drafted angles, and section views
9:00	Recommended Cutting Tools & Preferences	Flat end mills vs ball mills, corner radius tools, tool selection logic
18:00	Setup & Stock Considerations	Bounding box stock, alignment, masking surfaces, extra material
21:00	3-Axis Horizontal Roughing Demo	Offset pattern, step-over, stock to leave, avoiding parting plane
34:00	Step Minimization in Roughing	Combining roughing + re-roughing in one operation for efficiency
38:00	3-Axis Parallel Finishing (Pre-finishing)	Offset curve containment, X & Y direction passes, progressive tool sizes
52:00	Parallel Finishing (Final Passes)	Tight step-over (5%), tolerance settings, surface quality focus

Time	Topic / Section	Description
1:05:00	3-Axis Horizontal Finishing	When and why to use it for vertical walls/sidewalls
1:08:00	2½ Axis Profiling Discussion	Comparison and why it's often avoided for contoured parting planes
1:10:00	Final Parting Plane Cleanup	Corner radius mill + cleanup path for tangent fillets
1:25:00	Cut Material Simulation & Post-Processing	Full simulation, G-code output with arcs, wrap-up

Key Tips & Best Practices

Here are the most valuable practical tips from the webinar:

1. **Flat end mills for roughing** — They calculate faster and remove material more efficiently than ball mills. Only use ball or corner radius mills for roughing when you specifically need fillets.
2. **Step Minimization is a game-changer** — Set initial step-down at 50% of tool diameter, then enable step minimization at 25%. This combines roughing and re-roughing into one operation, saving significant time.
3. **Leave stock at the parting plane** — Never machine flat to the parting plane during roughing with a flat end mill. Leave ~0.025"–0.125" stock and finish it later with a ball mill or corner radius mill for clean tangent fillets.
4. **Parallel Finishing is usually faster than Horizontal Finishing**. Use it with offset curve containment and run passes in both X and Y directions (90° apart) for superior surface finish.
5. **Progressive tool sizes for finishing** — Start with larger ball mills (3/4") and step down to smaller ones (1/2" → 1/4") with tighter step-overs (down to 5%).
6. **Corner radius mill + cleanup path is excellent for final parting plane finishing** — it achieves tangent contact while clearing flats.
7. **Use masking surfaces** to protect holes and unwanted areas during roughing and finishing.
8. **Catch planes / containment** — Use offset curves or catch planes to control tool tangent at critical edges (especially important with different ball mill sizes).
9. **Tolerance settings**:

- Loosen during programming (e.g., 10,000) for faster calculation.
- Tighten for final output (e.g., 100 or 1,000).

10. **Avoid 2½ Axis Profiling on parts where contoured surfaces meet flat parting planes** — it cannot produce proper tangent fillets.
11. **Simulation best practice** — Use fine accuracy and distinct colors per operation. Pause and inspect critical areas (especially the parting plane).
12. **Post-processing** — Enable arc output (G2/G3) for smoother toolpaths. Default post-processor settings usually work well unless your machine requires specific adjustments.

Key Takeaways

This is an excellent foundational 3-axis webinar that teaches efficient, production-ready workflows for complex mold work. The biggest takeaways are:

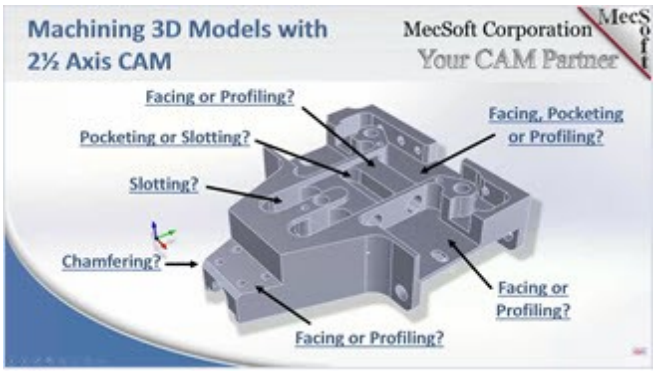
- Smart use of step minimization and parallel finishing in two directions dramatically improves efficiency and surface quality.
- Proper handling of the parting plane (leaving stock + dedicated cleanup) is critical for mold-quality results.
- Flat end mills + ball mills + corner radius tools each have specific roles — don't overuse ball mills in roughing.
- Prepare 3D geometry carefully before machining.
- Match cutting tools to the desired surface finish.
- Optimize roughing parameters to reduce cycle times.
- Use step minimization to improve motion efficiency.
- Adjust finishing stepover values for better surface quality.
- Combine 2½ axis and 3-axis operations strategically.
- Validate all toolpaths with simulation before machining.

4.6 2.5 Axis Advanced

Link: https://www.youtube.com/watch?v=8VCPlKtLXS4

This webinar shows how to efficiently machine complex 3D models using 2½-axis strategies (profiling, pocketing, facing, slotting, drilling, chamfering) in MecSoft CAM. It demonstrates the advantages of working directly from a 3D model rather than just 2D curves, with smart use of cut levels, entry/exit control, and trichoidal slotting.

Source Files: 20.25.07 AMS Webinar Machining 3D Models with 2½ Axis CAM.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Transition from curve-based to 3D model-based 2.5-axis machining
2:44	Geometry Requirements & Model Analysis	Part features (pockets, slots, chamfers), orientation, pre-planning curves
7:50	Setup and Stock Considerations	Bounding box stock, work zero alignment, pre-ground stock benefits
9:20	Profiling (Side Areas)	Open side machining with 19 mm flat end mill, step-over & cut levels
18:00	Facing (Open Pockets)	Island offset facing on open areas with 4 mm end mill
22:00	Pocketing (Roughing)	Closed pocket roughing with offset pattern
25:00	Profiling (Finishing Open Areas)	Edge finishing with 3 mm end mill, refined entry/exit control
33:00	Profiling with Offset (Clearing)	Using step-over in profiling to clear open regions efficiently
39:00	Drilling (Access Holes)	Peck drilling for slot entry points
41:00	Slotting (Trichoidal)	High-speed trichoidal slotting with ramp entry from drilled holes

Time	Topic / Section	Description
52:00	Profiling (Final Trimming)	External feature trimming with 10 mm end mill
58:00	Chamfering	45° chamfer using V-mill on front edge
1:00:00	Cut Levels, Simulation, Reports & Post-Processing	Depth control, full job simulation, setup sheets, G-code output

Key Tips & Best Practices Extracted

Here are the most valuable practical tips from the webinar:

1. **Leverage the 3D Model** — Use the actual 3D geometry for accurate top/bottom depths and cut sides instead of manually creating curves for every feature. This saves time and improves accuracy.
2. **Add Curves Strategically** — Even with a 3D model, add 2D curves where needed to precisely control toolpath boundaries, especially for complex or open features.
3. **Profiling with Offset is Powerful** — Use the "step over" feature in profiling operations to clear open areas efficiently without needing a separate pocketing operation.
4. **Trichoidal Slotting for High-Speed Machining** — Use trichoidal (high-speed) slotting with ramp entry from pre-drilled holes. Add a cleanup pass to reduce cusping.
5. **Control Entry/Exit Angles** — Set entry/exit to **zero angle** for clean, straight engagement in open areas. Use linear engagement (e.g., 4–14 mm) for better control.
6. **Pick Depth Accurately** — Use the "pick depth" tool on the 3D model for precise cut level definition instead of guessing.
7. **Facing for Open Pockets** — When traditional pocketing would leave unwanted fillets in open areas, switch to **Facing** with island offset for cleaner results.
8. **Drill First for Slot Entry** — Always pre-drill access holes before slotting to enable safe ramp entry and prevent tool damage.
9. **Cut Levels & "Distance as Specified"** — Set consistent depth per cut (e.g., 1–2.5 mm) and use "Distance as specified" to maintain uniform step-downs across operations.
10. **Tool Diameter Rules** — Always choose a tool smaller than the smallest feature constraint (especially for pocketing and profiling inside tight areas).
11. **Simulation After Each Operation** — Run cut material simulation frequently with distinct colors per operation to verify material removal and catch collisions early.

12. **Pre-Ground Stock Saves Time** — If stock is pre-machined/ground, you can skip top facing operations entirely.
13. **Reports & Setup Sheets** — Generate reports for tool lists, setup sheets, and estimated cycle times directly from the Machining Job.

Key Takeaways

This webinar is excellent for users who want to **maximize 2.5-axis efficiency** on 3D parts without jumping straight to full 3-axis contouring. The biggest lessons are:

- 2.5-axis strategies (especially profiling + offset and trichoidal slotting) can handle surprisingly complex 3D geometry when applied creatively.
- The 3D model itself becomes your best friend for depth control and feature recognition.
- Smart use of entry/exit control and pre-drilled access holes dramatically improves reliability and surface quality.
- Use 3D models to simplify setup and visualization
- Select cutting tools based on operation type and material removal goals.
- Combine standard 2½ axis operations for efficient machining workflows.
- Optimize cut levels to improve cycle time and tool life.
- Use simulation to verify setups before machining.
- Generate setup documentation to improve communication on the shop floor.
- This session bridges the gap between basic 2D curve machining and more advanced 3-axis work.

4.7 TURN Advanced

Link: https://www.youtube.com/watch?v=PRZfEihW6RA

This webinar teaches how to program turning operations directly from 2D drawings in the TURN module of MecSoft CAM (VisualCAD/CAM or RhinoCAM). It uses a real-world production part featuring a 2.5"-8 NPT tapered external thread, multiple inner diameters, an O-ring groove, and requires OD/ID turning, grooving, threading, Breakchip drilling, and parting off.

Source Files: 20.25.06 AMS Webinar Machining 2D Drawings in TURN.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of TURN module, part example, Q&A setup
2:30	Geometry Requirements for TURN	Closed 2D profile setup, origin alignment, tolerance settings (6 decimals)
5:00	Setup and Stock Considerations	Cylindrical stock definition, work zero at chuck face, extended back stock, machine limits
15:00	OD Turn Roughing and Finishing	Diamond insert (forward/backward), offset/linear cuts, split operations
25:00	GROOVE Roughing and Finishing	0.125" groove tool, punch in/out roughing, single-pass finishing
31:00	Breakchip Drilling (for ID roughing)	Stepped drills (0.5", 1.125", 1.732"), break increment settings, chip control
37:00	ID Turn Roughing and Finishing	ID forward tool, linear/offset cuts, clearance settings
44:00	Straight and Tapered Threading	2.5"-8 NPT external thread (G76 cycle), major diameter profile, spring passes, in-feed
52:00	Toolpath Simulation	Fine tolerance simulation after each operation, collision & limit checking

Time	Topic / Section	Description
56:00	Post-processing G-Code	Fanuc post (legacy mode), G76 threading output, tool change point setup
58:00	Conclusion & Q&A	Workflow recap, upcoming resources

Key Tips & Best Practices

Here are the most practical takeaways from the webinar:

1. **Profile Setup is Critical** — The 2D drawing must be a closed curve positioned at the XYZ origin with the center line on the X-axis and the back of the part on the Y-axis. Set CAD tolerances to 6 decimal places for accuracy.
2. **Extend Stock for Safety** — Add at least 0.5 inch of extra material beyond the back of the part. This prevents tool limit errors and allows safe parting off and tool overhang.
3. **Work Zero at Chuck Face** — Set the work zero at the rightmost face of the stock (chuck side) for consistent coordinate output.
4. **Split OD/ID Operations** — Use separate forward and backward tool orientations when geometry prevents a single pass (common on parts with back-face features).
5. **Breakchip Drilling for Deep IDs** — Use break chip drilling (0.1" increment) instead of turning when ID clearance is too tight for the tool holder. Step up drill sizes progressively.
6. **Thread Profile Must Match Major Diameter** — For external threads (especially tapered NPT), the 2D profile must represent the exact major diameter and taper angle. Use the thread specs to set pitch, depth, and in feed accurately.
7. **G76 Threading Cycle** — MecSoft's Fanuc post outputs clean single-line G76 cycles. Enable 2 spring passes (rough + finish) and constant volume in feed for best results.
8. **Clearance & Entry/Exit Angles** — Use 45° or -45° entry/exit angles with 0.062" approach/retract for safe tool engagement, especially on ID operations.
9. **Simulate After Every Operation** — Run simulation with fine tolerance after roughing, finishing, grooving, drilling, and threading. Enable "stop on error" to catch machine limit violations early.
10. **Groove Tool Width Matters** — A 0.125" groove tool works well for both roughing (punch in/out) and finishing in one or two passes.
11. **Machine Limits** — Always set realistic travel limits (±24" in the example) to match your actual lathe before simulating.

12. **Post-Processor Tip** — Use a legacy Fanuc-compatible post for best G76 threading output. Set tool change point (e.g., X10) for safe retracts.

Key Takeaways

This is one of the most practical TURN module webinars available. It shows a complete real-world workflow from 2D drawing to finished G-code, including the often-tricky combination of tapered threading + deep ID features + grooving.

Biggest strengths highlighted:

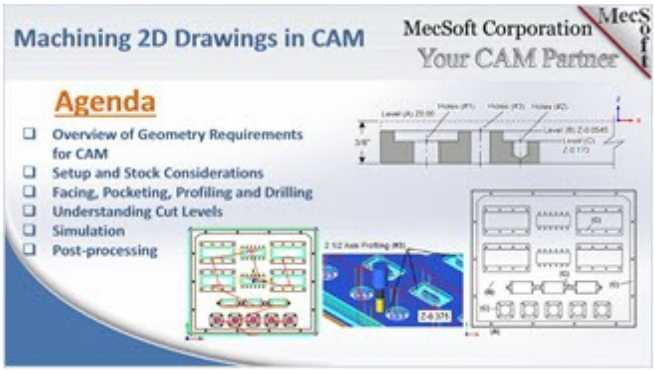
- Proper 2D profile setup and stock extension prevent most common errors.
- Breakchip drilling + ID turning hybrid approach solves tight-clearance problems.
- Clean G76 threading output with spring passes.
- Prepare clean and connected 2D profiles before machining.
- Use facing operations first to establish accurate reference surfaces.
- Optimize roughing depths to improve insert life and stability.
- Verify tool clearances carefully before posting code.
- Use simulation to validate TURN operations before machining.
- Combine roughing and finishing strategies for better efficiency.

4.8 2.5 Axis & 2D Dwgs

Link: https://www.youtube.com/watch?v=K8jdmr8h93s

This is a foundational webinar focused on programming directly from 2D drawings (no 3D model required) in MecSoft CAM (VisualCAD/CAM or RhinoCAM). It uses a real-world instrument panel prototype machined from 3/8" aluminum plate and covers the complete beginner-to-intermediate workflow: facing, pocketing, profiling, drilling, and tapping.

Source Files: 20.25.05 AMS Webinar Machining 2D Drawings in CAM.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Focus on 2D drawing workflows for new users
2:10	Geometry Requirements for CAM	2D curve preparation, merging curves, predefined regions, importing elevation view as background image
13:30	Setup and Stock Considerations	Box stock definition, work zero alignment, stock color preferences, ignoring wireframes
17:00	Facing	Top facing to part surface using island offset pattern
23:00	Pocketing (Perimeter Lip)	Closed pocket with cleanup pass, ramp entry
30:00	Profiling (Outer & Internal Cutouts)	Multiple profiling operations with step-over roughing + finishing
50:00	Drilling & Tapping	Peck drilling, break-chip drilling, peck tapping for 10-24 threads
1:05:00	Cut Levels, Simulation & Post-Processing	Depth management from 2D drawings, full simulation, G-code output

Key Tips & Best Practices Extracted

Here are the most useful practical takeaways from the webinar:

1. **Use a Background Image for Depths** — When working from pure 2D drawings, import an elevation view (PNG) via Modeling Aids > Background Image to visualize and accurately determine Z-depths.
2. **Predefine Regions for Fast Selection** — Create and save named curve regions (especially for hole groups) so you can quickly select multiple holes or pockets without re-selecting every time.
3. **Align Stock to Top of Geometry** — Set stock Z=0 at the top of the part for easier depth referencing when the drawing doesn't contain 3D information.
4. **Cleanup Pass in Pocketing** — Always enable the cleanup pass in pocketing operations to remove leftover stock in corners and ensure a clean finish.
5. **Ramp Entry is Safer** — Use ramp entry (instead of plunge) for pocketing and profiling, especially in aluminum, to reduce tool stress.

6. **Combine Roughing + Finishing in Profiling** — Use step-over (e.g., 0.100") with a larger tool to rough and finish in one operation when tolerance allows.
7. **"At Top" Cut Geometry Location** — When facing or starting from the top of stock, set cut geometry location to At Top for correct depth calculation.
8. **Predefined Hole Groups Save Time** — Group holes by size/type (e.g., all 1/4" through holes) so you can apply the same drilling parameters in one click.
9. **Peck Drilling & Break-Chip Settings** — For deep holes, use peck drilling with 0.1" increments and enable "Add toe tip" for full penetration on through holes.
10. **Tap Depth Calculation** — Start tap operations from the correct Z level (after facing + pocketing) and calculate tap depth to the bottom of the threaded hole (not full stock thickness).
11. **Tolerance = 0.001" for Finishing** — Use 0.001" tolerance when you want the toolpath to finish exactly to the geometry.
12. **Simulation is Essential** — Always simulate after each major operation (especially before drilling/tapping) to verify depths and avoid crashes.

Key Takeaways

This webinar is the perfect starting point for anyone new to MecSoft CAM or anyone who frequently works from 2D drawings (common in sheet metal, plates, and prototypes).

Biggest lessons:

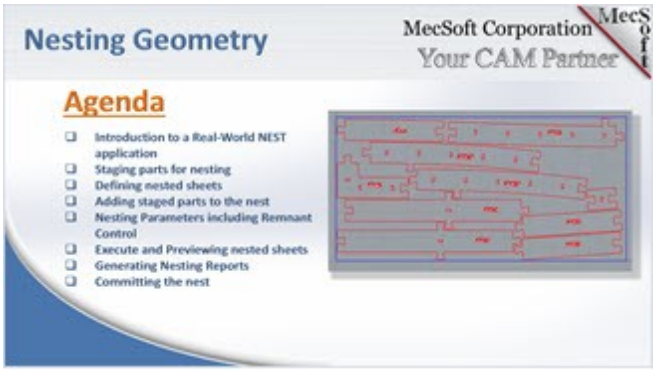
- Depths are not automatic in 2D drawings — you must carefully manage cut levels using background images or manual measurement.
- Predefined regions and smart stock setup dramatically speed up programming.
- The same 2.5-axis toolpaths (facing → pocketing → profiling → drilling/tapping) form the foundation for almost all plate and panel work.
- Clean and organize 2D geometry before machining.
- Use layers to simplify operation management.
- Apply lead-ins and lead-outs for improved edge quality.
- Automate drilling workflows with hole recognition.
- Optimize cut order to reduce machining time.
- Validate all toolpaths with simulation before posting code.

4.9 Nesting Real World

Link: https://www.youtube.com/watch?v=M7xKjNV6qh4

This webinar demonstrates the NEST module (included with RhinoCAM and VisualCAD/CAM for AMS subscribers) using a real-world production example from Road Yacht Design. It shows how to nest 24 complex 2D jig/fixture components for a massive 460 ft yacht assembly, achieving significant material savings through true-shaped nesting and smart remnant control.

Source Files: 20.25.03 AMS Webinar Nesting Real World Applications.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Real-World Case Study	Overview of Road Yacht Design jig/fixture project and NEST module benefits
3:07	Nesting Module Overview & True-Shaped vs Rectangular	Explanation of true geometry nesting advantages
7:18	Staging Parts for Nesting	Exploding geometry, adding parts to browser, preserving relationships
9:03	Defining Nested Sheets	Adding 12×5 ft sheets, thickness settings, work zero placement
11:30	Adding Staged Parts & Quantities	Setting part counts, priorities, fixed orientation, grain direction
19:28	Nesting Parameters & Remnant Control	Clearance, accuracy, tagging, layout options, remnant width constraints
29:01	Execute Nest & Preview Results	Estimating sheets, running nest, comparing true-shaped vs rectangular

Time	Topic / Section	Description
35:00	Commit Nest & Layer Management	Committing to separate layers, projecting to XY, remnant integration
39:13	Remnant Discussion & Optimization	Clean cut vs stepped remnants, labeling, reuse strategies
44:49	Rectangular Nesting Comparison	Side-by-side efficiency comparison (6 vs 9 sheets)
51:05	Machining Workflow After Nesting	Linking nests to toolpaths, consistent work zeros across layers
55:04	Conclusion & Resources	Blog post announcement, support contact, key takeaways

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **True-Shaped Nesting is Far Superior** — It uses actual part geometry instead of bounding boxes, dramatically reducing sheet count (in this case from 9 sheets down to 6).
2. **Fix Critical Orientations** — Use “Fixed Orientation” for long or structurally important parts. This single change reduced the sheet count from 7 to 6 in the example.
3. **Explode Everything Before Staging** — Grouped objects, blocks, and text must be exploded into individual curves. Labels must be converted to curves.
4. **Preserve Part Relationships** — Enable this option so inner cutouts stay attached to their parent parts instead of being treated as separate pieces.
5. **Remnant Control is Powerful** — Set a minimum width (e.g., 3") to filter out unusable scraps. Use “Clean Cut” for simple horizontal/vertical remnants or “Stepped Area” for maximum accuracy.
6. **Label Remnants Externally** — Always label remnants **outside** the geometry (above the sheet) so they don’t interfere with nesting and are easy to track in the warehouse.
7. **Stacked Layout + Separate Layers** — Use stacked layout and commit each sheet to its own layer. This keeps one consistent work zero for the entire job, making CNC programming much easier.
8. **Grain Direction & Priority** — Set grain direction (X or Y) when material has directional properties. Use priority settings to ensure critical parts are placed first.

9. **Clearance Matters** — Set realistic clearance (e.g., 0.5") based on your cutter diameter to prevent parts from touching after cutting.
10. **Remnants Can Be Reused** — After committing, remnants can be added back into the parts list for future nesting jobs, significantly reducing waste over time.
11. **Accuracy Setting** — Use "Low" accuracy for faster nesting calculations when final part geometry is not affected.
12. **Work Zero Consistency** — Place work zero at the lower-left corner of every sheet for predictable CNC setup.

Key Takeaways

This is one of the most practical **real-world nesting** webinars available. It proves that the NEST module isn't just theoretical — it delivers measurable results on large, complex projects (in this case saving multiple full sheets of plywood for a 460-ft yacht build).

Biggest wins highlighted:

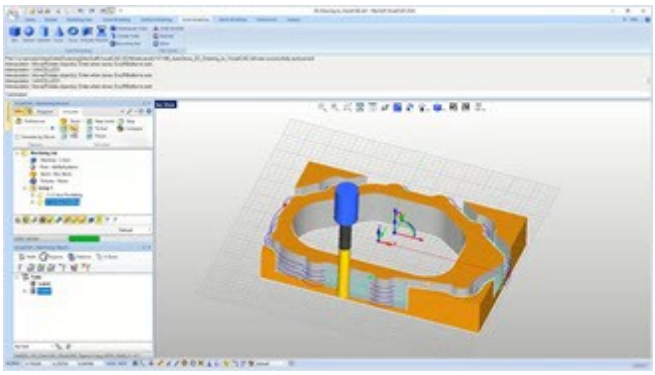
- True-shaped nesting + fixed orientation = major material savings
- Smart remnant control turns waste into usable stock
- Layer-based committing simplifies downstream machining
- Organize and clean geometry before nesting.
- Standardize sheet sizes to simplify production workflows.
- Control part rotation to preserve grain direction and orientation.
- Use remnant management to maximize material utilization.
- Preview nested layouts before generating toolpaths.
- Optimize spacing and nesting parameters for better sheet efficiency.
- The NEST module is included free with active AMS subscriptions for both RhinoCAM and VisualCAD/CAM.

4.10 VisualCAD/CAM

Link: https://www.youtube.com/watch?v=tWY6tm5nA-o

This is a foundational "getting started" webinar for VisualCAD/CAM. It covers the key differences from Rhino, file conversion/import workflows, basic modeling tools, and practical 2-axis & 3-axis CAM programming — ideal for new users or shop-floor teams transitioning to VisualCAD/CAM.

Source Files: 20.24.09 AMS Webinar VisualCADCAM Topics.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals, chat/Q&A, video distribution info
2:30	Differences Between Rhino and VisualCAD	Shop-floor focus vs advanced engineering modeling
7:30	Conversions to VisualCAD & Translators	RhinoCAM/SolidWorks export (.VCP), STEP/IGES import, orientation tools
14:00	Transform 2D Drawings & Measure/Analyze Geometry	DXF import, curve merging, measurement tools, stock bounding box
25:00	Modeling from DXF Files & Basic Modeling	Extrude curves, surface creation, trimming, filleting, section curves
35:00	VisualCAD Options	System/display settings, tolerance controls, grid & view preferences
38:00	2-Axis CAM Programming	DXF import → pocketing & profiling with ramp entry
50:00	3-Axis CAM Programming	STEP import, horizontal roughing, pocketing, profiling, spiral finishing
1:05:00	Resources & Support	Online help, VisualSERVE portal, AMS CAMJam videos, technical support

Key Tips & Best Practices

Here are the most practical takeaways from the webinar:

1. **VisualCAD is Shop-Floor Focused** — Use it for simpler geometry (DXF files, 2D machining, laser/plasma) rather than complex parametric engineering work. Rhino is better for advanced modeling and automation.
2. **Native .VCP Export from RhinoCAM** — Use the built-in export utility to transfer complete jobs (geometry + toolpaths) native into VisualCAD with zero data loss.
3. **Merge Curves Early** — Always merge curves after importing DXF files before extruding or creating toolpaths for clean, continuous profiles.
4. **Use Background Images or Section Curves** — When working from 2D drawings, import elevation views or create section curves to accurately determine depths and internal features.
5. **Stock Bounding Box Tool** — Quickly determine part size and verify dimensions before programming.
6. **Align Stock to Top of Geometry** — Set Z=0 at the top of the part for consistent and intuitive depth management.
7. **Ramp Entry/Exit in 2-Axis Operations** — Use ramp entry instead of plunging when geometry allows, especially on aluminum or harder materials.
8. **Predefined Regions for Efficiency** — Create and save named regions for repeated features (pockets, hole groups) to speed up programming.
9. **Tight Tolerances for Accuracy** — Use tight geometry tolerance settings during import and modeling; loosen curve hookup tolerance only if merging fails.
10. **Layer Management** — Organize geometry on separate layers (e.g., red for visibility) to keep complex jobs manageable.
11. **Simulate After Every Operation** — Always run cut material simulation to verify toolpaths, especially when switching between 2-axis and 3-axis strategies.
12. **Orientation Utility** — After importing STEP/IGES files, use the orientation tool to quickly align the part with Z upward for standard machining setup.
13. **Spiral Finishing for Better Surface Finish** — Use spiral finishing with small step-over (e.g., 0.1") and a ball mill for final passes on contoured areas.

Key Takeaways

This webinar is the ideal starting point for anyone new to VisualCAD/CAM. It clearly explains:

- When to choose VisualCAD vs Rhino
- How to efficiently import and prepare 2D drawings and 3D models

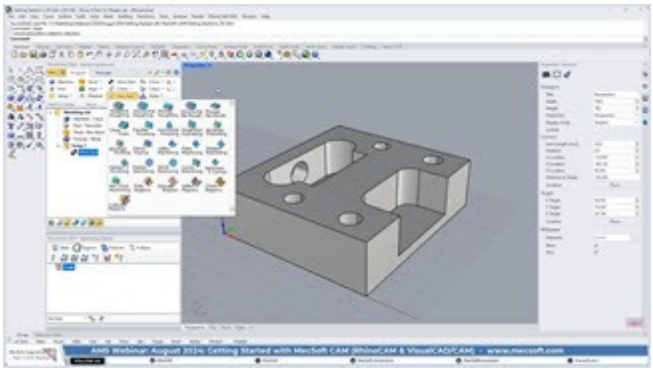
- Practical 2-axis and 3-axis workflows from real examples
- Clean and organize CAD geometry before machining.
- Use layers to separate machining strategies and simplify programming.
- Prepare geometry carefully using CAD editing tools.
- Reuse organized geometry across multiple machining operations.
- Verify all toolpaths with simulation before posting code.
- Use visualization tools to improve workflow clarity and setup accuracy.
- It also highlights the strong interoperability between RhinoCAM and VisualCAD/CAM — a major advantage for companies with both engineering and shop-floor teams.

4.11 Getting Started

Link: https://www.youtube.com/watch?v=U9Lv9xnGcW4

This is the ultimate beginner's guide to MecSoft CAM (RhinoCAM and VisualCAD/CAM). It walks new users through download, installation, licensing, interface basics, and programming their very first 2D and 3D parts — highly recommended for anyone just getting started.

Source Files: 20.24.08 AMS Webinar Getting Started with MecSoft CAM.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview for new users
0:00	Download & Installation	Where to download, installer tips, Rhino vs VisualCAD/CAM installation
10:00	License Activation & Demo Mode	How to activate, demo limitations, configuration levels (Standard/Expert/Professional)

Time	Topic / Section	Description
20:00	Interface Overview	Machining Browser, Tools/Regions tabs, module switching (Mill/Turn/Nest)
35:00	Programming Your First 3D Part	Stock setup, Work Zero, Horizontal Roughing & Finishing
50:00	Programming Your First 2D Part	DXF import, curve merging, pocketing & facing from 2D drawings
1:00:00	Support Resources	Online Help, VisualSERVE Portal, AMS CAMJam videos, requesting technical support

Key Tips & Best Practices

Here are the most important practical tips from the webinar:

1. **White-list MecSoft Emails** — Add mecsoft.com, sales@mecsoft.com, and support@mecsoft.com to your safe senders list to avoid missing download links and license codes.
2. **Run Installer as Administrator** — Always install as administrator and use the default installation path for easiest access to help files and assets.
3. **Save All Purchase Emails** — They contain your license code, download links, and Customer ID (starts with "MS") — critical for support.
4. **Demo Mode Limitations** — You can explore all features in demo mode, but you cannot post G-code or save operations. Disable internet for higher configuration access in demo.
5. **Work Zero is Critical** — Always set Work Zero at the machine's home position (usually top-left or top of stock). Incorrect Work Zero is the #1 cause of crashes.
6. **Use "Copy Model Bounding Box" for Stock** — This is the fastest and most accurate way to define stock from a 3D model.
7. **Prefer 3D Models Over 2D Drawings** — 3D models automatically provide depths and boundaries. 2D programming requires manual depth input and is more error-prone.
8. **Merge Curves After DXF Import** — Always merge curves before creating toolpaths to avoid gaps and broken profiles.
9. **Simulate Every Operation** — Run cut material simulation after every roughing and finishing pass before posting G-code.

10. **Start with Default Parameters** — Use factory defaults for your first jobs, then tweak only when needed.
11. **Request Custom Post Processors** — Over 300 posts are included. Email support@mecsoft.com for any machine-specific post you need.
12. **Keep Software Updated** — Regular updates are required to maintain support eligibility and access the latest features.
13. **Use the Machining Browser Tabs** — Switch between **Program** (create operations) and **Simulate** tabs for efficient workflow.
14. **Customer ID for Support** — When contacting support, include your Customer ID (last part of your activation code) for faster assistance.

Key Takeaways

This webinar is the **best starting point** for anyone new to MecSoft CAM. It removes the intimidation factor by showing exactly what to do from the moment you receive your license email.

Biggest lessons:

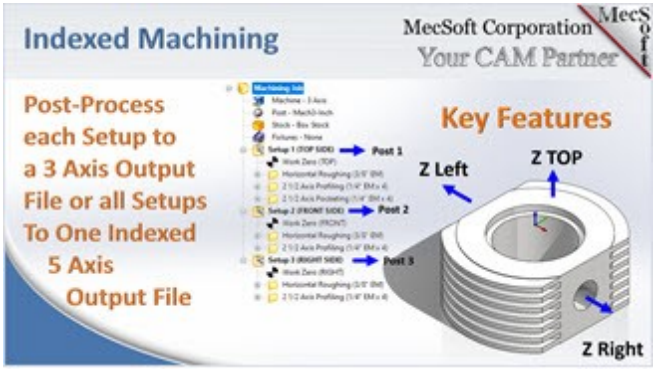
- Proper installation and licensing setup prevents 90% of early problems.
- Work Zero + Stock definition are the foundation of every successful job.
- Start simple (Horizontal Roughing + Finishing) and build from there.
- Take full advantage of the free resources (CAMJam videos, online help, VisualSERVE portal).
- Verify installation and licensing before starting CAM projects.
- Understand your CAM configuration and available modules.
- Use Quick Start Guides and tutorials to accelerate learning.
- Begin with simple 2D and 3D machining projects.
- Organize license information for easier management.
- Use the customer portal and CAMJam videos for ongoing learning.

4.12 5 Axis Indexed

Link: https://www.youtube.com/watch?v=lbq7xBGDQiQ

This webinar focuses on indexed machining (also called 3+2 or multi-sided machining) in RhinoCAM. It explains how to efficiently machine parts on multiple sides by locking rotary axes at fixed angles rather than using continuous 4/5-axis motion. Excellent for users with 4-axis or 5-axis machines who want simpler, more reliable programming.

Source Files: 20.24.07 AMS Webinar Indexed Machining in CAM.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:25	Introduction & Agenda	Overview of indexed machining from 3-axis to 5-axis
2:19	Coordinate Systems & Indexed Concepts	Machine vs Work Coordinate Systems, locking rotary axes
4:46	3-Axis Indexed: Flip Machining & Instancing	Two-sided parts, polar/XY instancing, guitar body example
8:38	Fixture Offset Programming	Using G54/G55 offsets for multiple parts or setups
10:14	4-Axis Indexed Machining (3+1 / Rotary Indexing)	Locking 4th axis, rotary instancing for radial features
12:31	5-Axis Indexed Machining (3+2)	Multi-sided machining with locked A/B axes
15:55	Post-Processor Considerations	Machine definition, output modes, tool change points, G-code output
48:00	Live Demonstrations & Best Practices	Real examples of flip machining, instancing, and multi-setup posting

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Use Setups for Indexed Machining** — Create separate setups for each orientation (top, bottom, side, etc.). This keeps everything organized and ensures correct work coordinate output.
2. **Instancing Saves Huge Time** — Instead of reprogramming the same features on every side, program once and use **XY Instance**, **Polar Instance**, or **Rotary Instance** folders. The post automatically generates all instances.
3. **Fixture Offsets for Multi-Part Setups** — Use G54, G55, etc., for each part or each indexed position. This is much cleaner than manually adjusting work zeros.
4. **Model Index Pins for Flip Machining** — When flipping parts, include index pins or dowels in your model for precise alignment between sides.
5. **Professional Configuration Required for Multi-Setup** — Only Professional and higher configurations allow multiple setups in a single file (essential for efficient indexed work).
6. **Post-Processor Must Support Indexed Output** — Make sure your post is set to "Output in Local Coordinate Systems" and correctly defines your machine's rotary axes (A on Z+, B on Y+, etc.).
7. **Always Post Entire Setups** — Posting a single operation may skip the rotary codes. Always post the whole setup or multiple operations together to get proper A/B axis output.
8. **Tool Change Point Safety** — Define a safe tool change point (e.g., X0 Y0 Z0 or machine home) in the post so the tool retracts before any rotary movement.
9. **Test Your Post Thoroughly** — New machines or 4/5-axis setups often require post adjustments. MecSoft offers paid post setup services if needed.
10. **3+2 is Often More Reliable Than Continuous 5-Axis** — For many parts, indexed (3+2) machining is simpler to program, easier to verify, and produces excellent results with less risk of collisions.
11. **Contact Positioning Control** — Use this feature in profiling operations to automatically stop the tool at the part perimeter instead of manually setting containment.

Key Takeaways

This webinar is excellent for anyone with (or planning to add) a **4-axis or 5-axis machine**. It shows how indexed machining can dramatically reduce programming time through smart use of **setups and instancing** while keeping G-code clean and reliable.

Biggest advantages highlighted:

- One program file can contain all sides/orientations
- Instancing eliminates repetitive programming
- Proper post setup ensures correct rotary codes and work offsets

- Use indexed machining to simplify multi-sided part programming.
- Prepare and orient models carefully before machining.
- Verify rotary setup orientation before generating toolpaths.
- Reuse familiar 3-axis machining strategies within indexed workflows.
- Validate indexed positions using simulation before posting code.
- Test rotary movements and post output before machining production parts.

4.13 Cabinet Automation

Link: https://www.youtube.com/watch?v=JvjHGgFMZyc

This webinar demonstrates a complete automation workflow for cabinet manufacturing using RhinoCAM. It shows how to go from a 3D cabinet assembly to fully nested, machined flat panels with minimal manual intervention — using Flat Panel Layout, the NEST module, Knowledge Bases, and Rhino scripting.

Source Files: 20.24.05 AMS Webinar Cabinet Automation.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Overview	Cabinet automation goals and workflow overview
3:00	Step 1: Cabinet Panel Layout	Flattening 3D assemblies into flat panels using Flat Panel Layout utility
9:00	Step 2: Nesting Panels on Sheets	True-shaped nesting with the NEST module, sheet setup, spacing rules
18:00	Step 3: Knowledge Base Creation	Predefining profiling and hole operations as reusable Knowledge Bases

Time	Topic / Section	Description
35:00	Applying Knowledge Base to Nested Sheets	Auto-generating operations on nested panels using layer rules
42:00	Advanced Automation via Scripting	Rhino + RhinoCAM API scripts for one-button geometry creation and operation generation
55:00	Conclusion & Best Practices	Workflow summary and scalability tips

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Use Consistent Layer Naming** — Name layers exactly the same in both your Knowledge Base template and production files (e.g., “outer” for perimeters, “inner” for holes). This enables automatic geometry selection.
2. **Set Cut Depth Slightly Oversize** — In Knowledge Bases, set total cut depth slightly deeper than nominal material thickness (e.g., 0.72" for 0.709" material) to accommodate minor thickness variations.
3. **Long Path Entry/Exit for Dense Nests** — Use long path entry/exit with overlap when profiling in tight nests to avoid tool collisions.
4. **Flatten Panels Before Nesting** — Use the Flat Panel Layout utility to convert 3D cabinet assemblies into flat poly surfaces on the XY plane. Ensure panels are merged solids/poly surfaces, not disconnected surfaces.
5. **True-Shaped Nesting is Ideal for Cabinets** — The NEST module handles irregular cabinet panel shapes far better than rectangular nesting, maximizing sheet utilization.
6. **Separate Sheets by Thickness** — Panels of different thicknesses cannot be nested together on the same sheet — the module will exclude mismatched parts.
7. **Knowledge Bases** = Massive Time Savings — Predefine profiling and hole operations once, then apply them instantly to any nested sheet with matching layer names.
8. **Scripting Takes Automation to the Next Level** — Use Rhino scripting + RhinoCAM API to automate geometry extraction (Duplicate Face Border), layer creation, and Knowledge Base application — turning a multi-step process into a one-click workflow.
9. **Helical Ramp Entry for Holes** — Use helical ramp entry with pitch-based depth control for clean, safe entry into circular cutouts.
10. **Test Knowledge Bases on Sample Files First** — Always verify your Knowledge Base on a test file before applying it to production nesting jobs.

11. **Maintain Topological Relationships** — When using the NEST module, keep parts as poly surfaces/solids so relationships (holes belonging to panels) are preserved.

Key Takeaways

This webinar shows a highly practical, real-world automation pipeline for cabinet shops:

- Organize cabinet geometry by machining operation type.
- Use layers and templates to automate repetitive workflows.
- Apply nesting strategies to maximize sheet utilization.
- Optimize machining sequences to reduce production time.
- Use labeling workflows to improve shop-floor organization.
- Validate cabinet layouts and machining operations with simulation.

Biggest benefits highlighted:

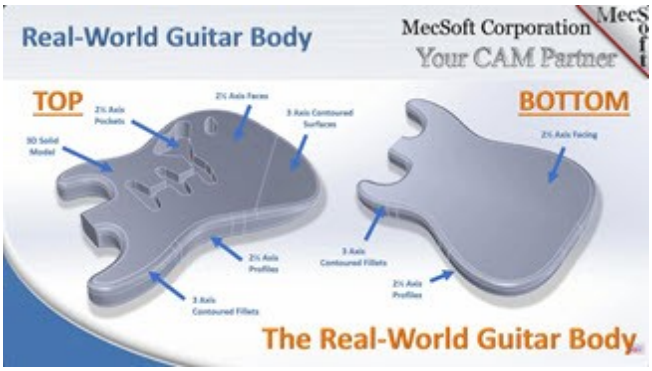
- Dramatic reduction in manual programming time
- Consistent, repeatable results across jobs
- Scalable from small shops to higher-volume production
- The combination of Flat Panel Layout + NEST + Knowledge Bases + Scripting creates a powerful cabinet automation system within RhinoCAM or VisualCAD/CAM.

4.14 Guitar Body

Link: https://www.youtube.com/watch?v=brptd6k_8Ic

This webinar demonstrates real-world 2.5-axis and 3-axis machining of a guitar body in RhinoCAM, including two-sided (flip) machining, pocket creation, sculpted surface finishing, and a detailed comparison between Standard and Professional configurations.

Source Files: 20.24.04 AMS Webinar Real World Guitar Body.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:30	Introduction & Agenda	Overview of guitar body project and configuration comparison
2:00	Part Setup and Fixturing	Stock modeling, work zero, two-register pin flip Fixturing
10:00	2.5-Axis Operations (Facing + Pocketing)	Face milling, pickup cavity pocketing, sidewall profiling
25:00	3-Axis Finishing (Horizontal Roughing + Parallel Finishing)	Contoured surface finishing with new 2024 “Contact Extent Condition”
40:00	Two-Sided (Flip) Machining Workflow	Top vs bottom file management in Standard configuration
50:00	Professional Configuration Advantages	3D Offset Pocketing, 3D Offset Profiling, multiple setups in one file, stock from selection
1:03:00	Estimated Machining Time & Conclusion	Operation timing, configuration recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Model a Stock Box for Flip Machining** — Create a stock box that fully contains the part. This guarantees perfect alignment when flipping the part and eliminates registration errors.
2. **Use Only Two Register Pins** — Over-constraining flip Fixturing with more than two pins can cause binding. Two well-placed pins are sufficient and more reliable.
3. **“Contact Extent Condition”** — In Parallel Finishing, enable this under Part Surfaces to ensure the tool reaches the full tangent line on fillets and contoured edges (solves previous version limitations).
4. **Clone Operations for Efficiency** — When creating similar pockets or profiles, clone the operation and simply reassign new control geometry instead of recreating everything from scratch.
5. **Offset Outer Curves for Extra Stock Removal** — When profiling around contoured areas, offset the outer curve outward by the tool diameter if you need additional material removal around fillets.

6. **Use "Stock from Selection" in Professional** — This powerful feature lets you include holes, alignment pins, and fixtures directly in the stock model for much more accurate simulation.
7. **3D Offset Profiling for Sculpted Surfaces** — In Professional configuration, 3D Offset Profiling maintains constant scallop height and follows complex curves beautifully — ideal for guitar bodies and organic shapes.
8. **Run Parallel Finishing in Two Directions** — Perform parallel finishing at 0° and 90° for superior surface finish on contoured parts.
9. **Enable Arc Fitting** — Use arc fitting (especially in 2.5-axis profiling) with a tolerance around 0.02× global tolerance to reduce G-code size and improve machine motion.
10. **Set Work Zero at Highest Z + Southwest Corner** — This consistent convention makes flip machining much easier and more predictable.
11. **Estimate Machining Time** — Right-click any operation or folder → Information to see estimated cycle time (very useful for quoting).
12. **Standard vs Professional:**
    - **Standard:** Great for most jobs but limited to one setup per file (requires separate top/bottom files for flip machining).
    - **Professional:** Allows multiple setups in one file, advanced 3-axis operations, and much better stock modeling.

Key Takeaways

This webinar is an excellent real-world case study that shows how to machine a complex sculpted part (guitar body) using both basic and advanced techniques.

Biggest lessons:

- Two-sided flip machining is very manageable with proper stock modeling and Fixturing.
- The Professional configuration unlocks significant time-saving and quality-improving features (especially 3D Offset operations and multi-setup files).
- The new "Contact Extent Condition" in 2024 makes parallel finishing much more reliable on contoured parts.
- Plan Fixturing and work holding before creating toolpaths.
- Optimize roughing strategies to reduce machining time and tool wear.
- Balance finishing stepover values against cycle time requirements.
- Use appropriate woodworking cutters to improve surface quality.
- Validate machining operations with simulation before cutting material.

4.15 Real-World Parts

Link: https://www.youtube.com/watch?v=SVF6zYp_2vs

This webinar demonstrates practical CAM programming of a real-world prismatic part with repetitive features (pockets, slots, and a sloped surface). It highlights efficient workflows using 2.5-axis and 3-axis operations, with a strong emphasis on XY Instancing in the Professional configuration.

Source Files: 20.24.03 AMS Webinar Real-World Parts.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:21	Introduction & Housekeeping	Webinar goals and invitation to submit real parts
2:20	Part Overview and Preparation	Part features, orientation, curve creation for control geometry
6:27	Software Configuration and Setup	Machine setup, post selection, stock & work zero definition
11:43	Pocketing Operations	Full-length slot pocketing with cleanup pass
29:00	Horizontal Roughing	Roughing with increased stock allowance near slopes
32:00	Horizontal Finishing	Corner radius finishing of vertical walls
40:00	Profiling (Cleanup)	Edge cleanup using bottom edges of sloped surface

Time	Topic / Section	Description
44:00	Between Two Curves (Professional Feature)	Finishing sloped surfaces using dual-curve control
47:00	XY Instancing for Repetition	Duplicating operations across 15" spaced features
51:00	Post Processing	Posting entire instance folders and tool priority options

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Orient the Part Exactly as It Will Sit on the Machine** — Match world X/Y to machine X/Y (long side along Y, short side along X) to avoid confusion during setup.
2. **Increase Stock Allowance Near Sharp Corners/Slopes** — Give the tool extra stock (e.g., 0.188") near tight corners or slopes to prevent the tool from wrapping around and crashing.
3. **Adjust Geometry Boundaries to Prevent Tool Wrapping** — When finishing near edges, shrink or offset control geometry so the tool stays at least ½ tool diameter inside the part edge.
4. **Use "Between Two Curves" for Sloped Surfaces** — This Professional feature is excellent for finishing angled or sculpted areas by controlling the toolpath between a top and bottom curve.
5. **Clone Operations Instead of Recreating Them** — When you need similar operations, clone them and simply reassign new control geometry — much faster than starting from scratch.
6. **XY Instancing is a Massive Time Saver** — In Professional configuration, create one set of operations and use an **XY Instance** folder to automatically duplicate them across repetitive features (e.g., 5 identical sets spaced 15" apart).
7. **Measure Spacing Corner-to-Corner** — When setting up instancing, measure from the same corner of one feature to the same corner of the next to ensure accurate spacing.
8. **Roughing Operations Can Be Used as Finishing** — With zero stock allowance and tighter tolerances, a roughing operation can serve as a finishing pass (very useful in Standard configuration).
9. **Use 3D Model to Determine Cut Side** — In profiling operations, let the 3D model automatically determine which side of the curve to cut on.

10. **Post Entire Instance Folders** — Post the whole XY Instance folder at once so the G-code is organized cleanly by instance (all operations for Set 1, then Set 2, etc.).
11. **Tool Priority Posting Option** — You can post operations grouped by tool number (ascending or descending) instead of tree order — helpful for minimizing tool changes.
12. **Standard vs Professional**:
    - **Standard**: You must manually duplicate operations for repetitive features.
    - **Professional**: Use XY Instancing and advanced features like "Between Two Curves" for dramatically faster programming.

Key Takeaways

This webinar is an excellent **real-world programming case study**. It shows how to efficiently handle parts with repetitive features and challenging geometry (slopes, tight corners) using both basic and advanced techniques.

Biggest lessons:

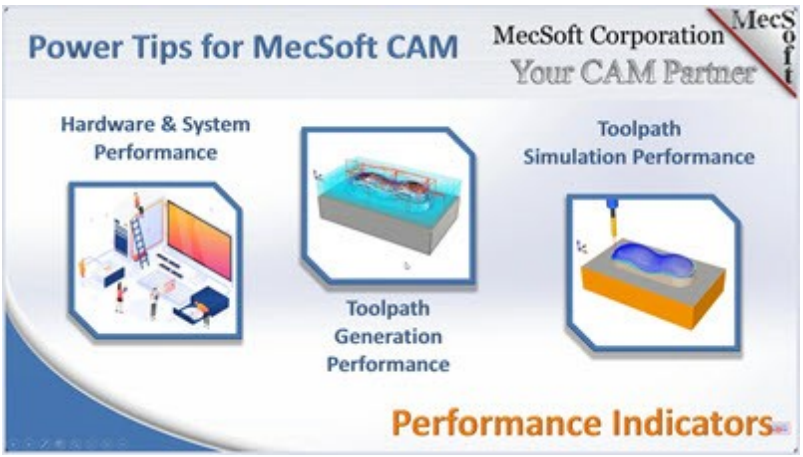
- Proper preparation (curves, stock, work zero) prevents most problems.
- XY Instancing (Professional) is one of the biggest productivity boosters for repetitive parts.
- Small adjustments to stock allowance and control geometry can prevent tool crashes and improve finish quality.
- Plan setups and machining orientation before creating toolpaths.
- Use facing operations to establish reliable machining references.
- Optimize depth cuts to improve tool life and machining stability.
- Apply advanced 3-axis strategies for complex geometry.
- Balance surface finish quality against machining cycle time.
- Verify machining operations using simulation before posting code.

4.16 Power User

Link: https://www.youtube.com/watch?v=NnQujxaJdzE

This is a highly practical "power user" session packed with advanced tips, hidden features, performance optimizations, and troubleshooting advice for RhinoCAM and VisualCAD/CAM users.

Source Files: 20.23.11 AMS Webinar Power Tips for MecSoft CAM.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of power tips topics
2:00	Dialing in Accuracy	Tolerance settings, arc fitting, internal vs displayed precision
15:00	Optimizing System Performance	Hardware recommendations, geometry types, toolpath patterns, multi-threading
35:00	Toolpath Properties (Hidden Features)	Right-click Properties dialog, custom G-code, color coding
40:00	Using G-Code Back plotters	Debugging G-code with MecSoft's editor and third-party tools
45:00	Post-processing Tips	Fixing missing posts after upgrades, custom post storage, Python posts
1:10:00	Navigating Licensing	Cloud vs Network Lock licenses, auto-release features

Key Power Tips & Best Practices Extracted

Here are the most valuable advanced tips from the webinar:

Accuracy Tips

1. Internal precision is 14 decimal places (display shows only 6) — use this when working with very tight tolerances.
2. Arc Fitting converts linear moves to G2/G3 arcs — dramatically improves surface finish and reduces file size, but increases segment count.
3. Set Arc Fitting Tolerance to 2× Global Tolerance for best results.
4. Match your post-processor decimal places (usually 5 or 6) to your tolerance settings.
5. Tighter tolerances = better finish but much slower calculation, simulation, and machining time.

Performance Optimization Tips

6. Hardware priorities: Fast CPU clock speed > more cores > SSD > 32GB+ RAM > dedicated GPU.
7. 2D geometry is fastest; dense meshes (from 3D scans) are slowest and can cause infinite loops.
8. Fastest toolpath patterns: Offset or Linear (avoid spirals when speed matters).
9. Fastest tools: Flat end mills for roughing; ball mills for finishing; avoid tapered mills unless necessary.
10. Enable “Generate toolpaths in multiple threads” in Preferences to use all CPU cores.
11. Reboot your computer daily and run only one Rhino/CAM session at a time.
12. Set Rhino priority to “Real Time” in Windows Task Manager for long calculations.
13. Use auto save during long toolpath generations.

Toolpath Properties (Hidden Gem)

14. Right-click any operation → Properties to:
    - Add custom comments or G-code snippets (use % prefix to output actual G-code).
    - Assign custom colors for easier simulation visualization (set Simulation tab to “MOP” mode).
    - View output file-name and last post date for job tracking.

G-Code Back plotting

15. Always backplot your G-code before running on the machine.
16. Use MecSoft’s G-Code Editor (included with AMS) or third-party viewers.
17. The toolpath viewer shows motions — you must post-process to see actual G-code.

Post-Processing Power Tips

18. If your post disappears after an upgrade:
    - Set it in a new file → Save Preferences to Registry, or
    - Save as a template and reload it.
19. Store custom posts in a separate folder (not the default location) to prevent them from being overwritten during updates.
20. Use the "Load from" field in Preferences to point to your custom post folder.
21. Python-based posts (introduced in 2023) offer much more flexibility for custom output.

Licensing Tips

22. Cloud License (1–3 seats): Auto-releases after 8 hours — great for travel or multiple machines.
23. Network Lock License (5+ seats): More secure cloud-based option.
24. Node-locked licenses are being phased out.
25. You can manually release a license if needed.

Key Takeaways

This webinar is a goldmine of productivity hacks for intermediate to advanced MecSoft CAM users. The biggest themes are:

- Balance accuracy vs performance — don't use tighter tolerances than your job actually requires.
- Leverage hidden features like Toolpath Properties and custom post storage.
- Optimize your system (hardware + settings) for faster calculations.
- Use back plotting and proper post management to avoid costly mistakes on the machine.
- Highly recommended for anyone who wants to get more speed and reliability out of RhinoCAM or VisualCAD/CAM.

4.17 4 Axis Advanced

Link: https://www.youtube.com/watch?v=_YyyQsRU2x8

This is an **advanced 4-axis training webinar** that goes beyond basic indexing. It demonstrates true continuous 4-axis techniques, drive surface machining, projection pocketing, and efficient 3+1 workflows on a complex real-world part.

Source Files: 20.23.09 AMS Webinar Advanced 4 Axis Machining.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of advanced 4-axis techniques
2:05	Part Setup and Machine Configuration	4-axis machine definition, stock setup, rotary axis orientation
6:00	3+1 Indexed 4-Axis Roughing	Top and bottom horizontal roughing with 180° rotation
14:00	2-Axis Chamfering + 3-Axis Finishing	V-mill chamfering and horizontal finishing
20:00	4-Axis Facing (Cylindrical Surface)	Facing between offset curves on a cylinder
26:00	4-Axis Engraving	Tool tip following curves on cylindrical features
29:00	4-Axis Profiling	Edge cleanup and material removal on cylinder ends
33:00	4-Axis Drive Surface Machining	True simultaneous 4-axis (all axes moving together)
43:00	Projection Pocketing	Machining grooves not normal to the rotary axis

Time	Topic / Section	Description
50:00	4-Axis Options, Safety & Best Practices	Auto-lift, tool change points, post-processor considerations

Key Tips & Best Practices Extracted

Here are the most valuable advanced tips from the webinar:

1. **Offset Curves by Tool Radius** — When facing or engraving on cylindrical features, always offset your control curves outward by the tool radius to prevent gouging into the main stock.
2. **Use Masking Surfaces** — Create masking surfaces to maintain stock connections during wrapping operations and prevent the tool from cutting into areas you want to keep.
3. **Drive Surface Machining = True 4-Axis** — This is one of the few operations where the tool, linear axes, **and** rotary axis all move simultaneously. It requires manually creating a drive surface (not a simple wrap).
4. **Projection Pocketing for Non-Normal Features** — When grooves or features are not perpendicular to the rotary axis (e.g., helical or skewed), use **Projection Pocketing** to project the toolpath onto the underlying surface while keeping the tool normal to the rotary axis.
5. **Auto-Lift During Rotation** — Enable the safety option to automatically lift the tool to the tool change position before the table rotates. This prevents collisions.
6. **Define Tool Change Point** — Set a safe tool change point in General Parameters so the post automatically inserts a retract move between setups.
7. **Engraving Only Works When Curves Are Normal** — 4-axis engraving assumes the curves are normal (perpendicular) to the rotary axis. For skewed features, use Projection Pocketing instead.
8. **Verify Post-Processor Support** — Not all posts handle advanced 4-axis motion correctly. Test thoroughly or request a custom post from MecSoft support.
9. **Simulate Every Operation** — Especially important in 4-axis work — check tool orientation and collision detection after every setup change.
10. **Flute Length Matters** — Make sure your tool has enough flute length to reach the widest material section when the part is rotated.
11. **Part Files Available** — The webinar provides down loadable part files (link in the video description) so you can follow along and practice.

Key Takeaways

This webinar is excellent for users who want to move beyond basic 3+1 indexing into **true continuous 4-axis machining**. The standout techniques are:

- Use indexed machining when simultaneous rotary motion is unnecessary.
- Align rotary axes and models carefully before machining.
- Optimize stepover values for improved rotary surface finish.
- Validate rotary movement using simulation before posting code.
- Monitor machine limits and rotary travel carefully.
- Apply roughing and finishing strategies appropriate for cylindrical geometry. Drive Surface Machining — For complex simultaneous 4-axis motion
- Projection Pocketing — For features not aligned with the rotary axis
- Smart use of offset curves and masking — To protect stock and achieve clean finishes
- Best for: Users with 4-axis machines who want to machine more complex geometries efficiently while maintaining good surface finish and tool life.

4.18 3 Axis Basic

Link: https://www.youtube.com/watch?v=k5jWd-mLiaI

This AMS webinar focuses on the fundamentals of basic 3-axis machining in RhinoCAM and VisualCAD/CAM. The presentation walks through essential workflows used to machine 3D parts using standard 3-axis operations, including setup preparation, stock definition, roughing strategies, finishing toolpaths, cutter selection, simulation, and post processing. Throughout the webinar, the presenters also share practical machining tips and beginner-friendly recommendations for improving workflow efficiency, reducing setup errors, and achieving better surface finishes when programming 3-axis CNC machining operations.

Source Files: 20.23.07 AMS Webinar Basic 3Axis Machining.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of basic 3-axis strategies and parts
2:47	Part Overview and Approach	Mold insert and Thermoform mold core examples
9:37	Part Setup, Tools & Stock	Machine definition, tool libraries, stock & work zero
19:02	3-Axis Horizontal Roughing	Z-level roughing with masking surface
34:07	3-Axis Horizontal Finishing (Parallel)	Ball mill finishing with tight tolerances
43:43	3-Axis Spiral Machining	Inside-out spiral finishing
45:29	3-Axis Radial Machining	Radial zigzag pattern from center
47:00	2-Axis Pocketing	Pocket clearing for bosses and features
53:07	Pencil Trace & Valley Machining	Edge cleanup (Professional feature)
55:18	Horizontal Finishing as Pencil Trace Alternative	Achieving similar results without Professional Config
59:42	Simulation, Post-Processing & Wrap-Up	Toolpath verification and final notes

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Use Masking Surfaces** — Create a masking surface (by revolving a cross-section) to protect pockets, bosses, and edges during finishing operations. Flip the surface normal if the toolpath doesn't behave as expected.
2. **Helical Engagement is Safer** — Always use helical ramp entry for roughing and pocketing to reduce tool stress and prevent crashes.
3. **Tight Tolerances for Finishing** — Use 0.001" (1 thou) or tighter tolerance for finishing passes to achieve good surface finish.

4. **Arc Fitting = 2× Global Tolerance** — Set arc fitting tolerance to twice your global tolerance for best results.
5. **Offset Perimeter Curves** — When finishing near edges, offset the control curve outward by tool radius + clearance to keep the tool from running off the part.
6. **Merge Curves Before Machining** — Always merge curves used for containment or profiling to avoid open loops and errors.
7. **Clear Flats in Finishing** — Enable "Clear Flats" in horizontal finishing to ensure the tool stays on the surface rather than lifting unnecessarily.
8. **Spiral & Radial Machining Tips:**
    - Great for contoured cavities.
    - Do not support arc fitting (only works in principal planes).
    - Use lower tolerance to compensate for the lack of arcs.
9. **Pencil Trace Alternative Without Professional** — Use Horizontal Finishing with "Clear Flats" and a single cut level to achieve similar edge cleanup results on flat areas.
10. **Stock Allowance Strategy** — Leave 0.010"–0.025" stock after roughing for finishing passes.
11. **Part Files Available** — Download the demonstration part files from the link in the video description to follow along.
12. **Work Zero Consistency** — Set work zero at the top-left (highest Z, southwest) corner of the stock for predictable results.

Key Takeaways

This webinar is an excellent hands-on introduction to the core 3-axis strategies you'll use most often:

- Horizontal Roughing → Bulk material removal
- Horizontal Finishing (Parallel) → General surface finishing
- Spiral / Radial → Efficient finishing of contoured cavities
- Pocketing + Profiling → Feature machining
- Pencil Trace (or alternative) → Edge and fillet cleanup
- Best for: Beginners to intermediate users who want to build a solid foundation in 3-axis mold and tooling work.
- Prepare clean and organized geometry before machining.
- Match cutting tools to roughing and finishing objectives.
- Optimize roughing operations before refining finishing strategies.

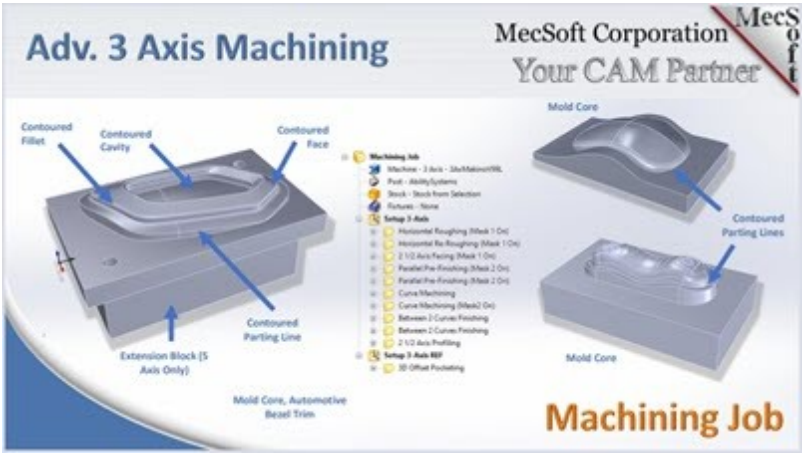
- Balance surface quality against machining cycle time.
- Use simulation to validate machining workflows before posting code.
- Define stock and work zero locations carefully for reliable setups.

4.19 3 Axis Advanced

Link: https://youtu.be/QmM-VSn4G7w

This August 2023 AMS Webinar Replay from MecSoft Corporation focuses on **Advanced 3-Axis Machining** strategies in RhinoCAM and VisualCAD/CAM (Professional configuration). Using a complex automotive bezel trim core mold part, the session demonstrates practical techniques for roughing, pre-finishing, and finishing challenging geometry. It covers part setup with Stock from Selection, Horizontal Roughing/Re-Roughing, Parallel Pre-Finishing with masking, Curve Machining, Between 2 Curves, 3D Offset Pocketing, Valley Re-Machining, Pencil Trace, plus 2-axis operations, 5-axis previews, tapered tools for draft angles, and cut material simulation. The goal is to show how advanced toolpaths handle areas that standard strategies struggle with on intricate mold cores.

Source Files: 20.23.08 AMS Webinar Advanced 3 Axis Machining.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Part Overview	Webinar goals, part introduction (automotive bezel trim core mold), and why advanced 3-axis strategies are needed for complex geometry.
1:00	3 Axis Part Setup & Stock from Selection	Defining stock by selecting a bounding box, setting work zero, and hiding stock for cleaner views.

Time	Topic / Section	Description
2:30	3 Axis Horizontal Roughing	Large end mill roughing with automatic stock removal calculation and default parameters.
4:00	3 Axis Horizontal Re-Roughing	Removing remaining material with smaller tools after simulating previous operations to update in-process stock.
6:30	2 Axis Facing & Profiling	Facing flat areas and profiling at the parting line with ramp entry and controlled retracts.
9:00	3 Axis Parallel Pre-Finishing	Using masks for holes and extended parting surfaces; running in X and Y directions with tight step-over.
14:00	3 Axis Curve Machining	Following curves projected onto surfaces within a bandwidth for wall cleanup without touching walls.
18:00	3 Axis Between 2 Curves Machining	Morphing toolpaths between two curves for consistent step-over on contoured/filleted surfaces.
23:00	3D Offset Pocketing	Professional-level pocketing that offsets directly on the surface for uniform cusp height on complex geometry.
27:00	3 Axis Valley Re-Machining	Advanced finishing for bi-tangent surfaces using reference tool diameter to calculate contact points automatically.
32:00	3 Axis Pencil Trace Machining	Single-pass bi-tangent finishing along edges, often used with tapered tools for mold details.
36:00	5 Axis Toolpaths Preview	Brief demonstrations of 5-axis Between 2 Curves, Swarf, and Curve machining for superior control on complex molds.
40:00	Tapered Tools & Draft Angle Machining	Using tapered ball mills matched to part draft angles to machine draft and finish in one operation.
46:00	Cut Material Simulation & Toolpath Viewing	Multiple simulations, visualization settings, and best practices for reviewing results throughout the process.

Time	Topic / Section	Description
52:00	Conclusion, Tips & Resources	Key takeaways, masking/parting line strategies for molds, part file access, and additional learning resources.

Key Tips & Best Practices Extracted

1. **Use Stock from Selection:** Quickly define accurate stock by selecting a bounding box model instead of manual measurements for faster and more precise setup.
2. **Simulate Before Re-Roughing:** Always run cut material simulation on previous operations so the software can calculate remaining stock accurately for re-roughing passes.
3. **Apply Masking Surfaces:** Use masks to protect holes, pockets, and EDM areas while extending parting line surfaces to prevent tool rollover and control boundaries.
4. **Run Parallel Pre-Finishing in Both Directions:** Machine in X then Y (or vice versa) with tight step-over for more uniform material removal and better surface preparation.
5. **Set Arc Fitting Tolerance Higher:** Make arc fitting tolerance at least double the machining tolerance for smoother toolpaths and better machine performance.
6. **Prefer Between 2 Curves on Contoured Surfaces:** Use this strategy instead of parallel finishing when you need consistent step-over width regardless of surface angle or curvature.
7. **Leverage Valley Re-Machining for Bi-Tangent Areas:** Let the software automatically calculate bi-tangent contact points using a reference tool diameter for superior finishing in mold corners and valleys.
8. **Match Tapered Tools to Draft Angles:** Use tapered ball mills with the same angle as the part's draft to create draft and finish the surface in a single operation.
9. **Control Retracts at Parting Lines:** Use ramp entries, no-retract settings, or positive stops to avoid dragging the tool across finished parting surfaces.
10. **Choose Largest Practical Ball Mill for 3D Offset Pocketing:** Bigger tools with this operation create more uniform cusp heights across complex contoured and vertical surfaces.

Key Takeaways

- Advanced 3-axis strategies in RhinoCAM/VisualCAD/CAM (Professional) enable precise machining of complex mold geometry that standard operations cannot handle well.
- Proper part setup with Stock from Selection and strategic masking is essential for controlling toolpaths on intricate mold cores.
- Horizontal Roughing followed by Re-Roughing (after simulation) efficiently removes bulk material before moving to finishing operations.

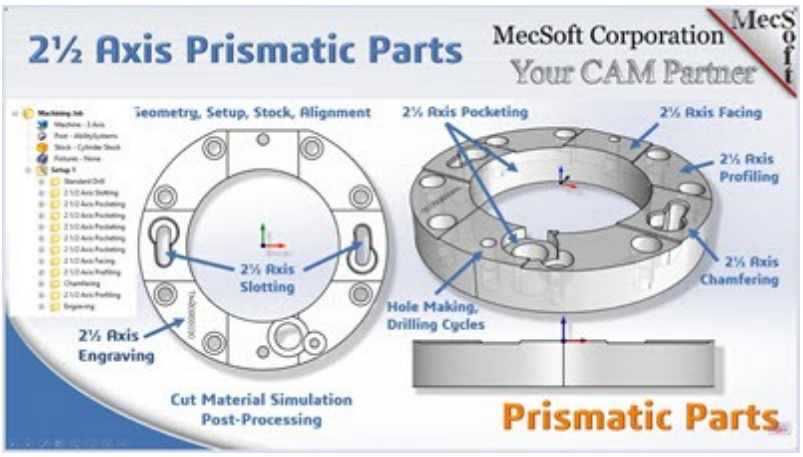
- Between 2 Curves and 3D Offset Pocketing provide more consistent results than traditional parallel finishing on curved or contoured surfaces.
- Valley Re-Machining and Pencil Trace are powerful specialized tools for cleaning up bi-tangent edges and mold details with high precision.
- Tapered tools matched to draft angles allow combining draft creation and finishing into one efficient operation.
- Simulation and careful toolpath visualization are critical for verifying results, especially when working with in-process stock and advanced strategies.
- These techniques work best when combined with good CAD practices like untrimming surfaces and consistent parting line management.

4.20 2.5 Axis Prismatic

Link: https://www.youtube.com/watch?v=xhRHJStrqw4

This AMS webinar focuses on machining prismatic parts using 2½ Axis machining strategies in RhinoCAM and VisualCAD/CAM. The presentation explains how common real-world mechanical components composed of flat faces, pockets, holes, and vertical walls can be machined efficiently using standard 2½ Axis operations. Topics include geometry preparation, stock setup, cut levels, facing, pocketing, profiling, drilling, simulation, and post processing. Throughout the webinar, the presenters also share practical machining tips for improving programming efficiency, reducing cycle time, and producing accurate prismatic parts using proven CAM workflows.

Source Files: 20.23.05 AMS Webinar 2½ Axis Prismatic Parts.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Part Overview	Circular prismatic part with multiple 2.5-axis features
0:00	Geometry Definition	Adding 2D curves (lines, circles) for control geometry
10:00	Setup, Stock & Alignment	Stock definition, work zero, single-setup workflow
15:00	Hole Making & Drilling Cycles	Spot drilling, deep drilling, center drilling
25:00	2.5-Axis Facing	Facing recessed areas with negative stock for clean edges
30:00	2.5-Axis Pocketing (High-Speed + Multi-Depth)	Spiral pocketing, handling varying pocket depths in one operation
40:00	2.5-Axis Slotting	Machining slots using spine curves (when tool = slot width)
50:00	2.5-Axis Profiling (Step-down & Outer)	Profiling stepped areas and outer perimeter
55:00	2.5-Axis Chamfering	V-bit chamfering using bottom edges
1:00:00	2.5-Axis Engraving	Text engraving with V-bit and single-stroke fonts
1:05:00	Simulation & Post-Processing	Cut material simulation and G-code output

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Negative Stock Allowance for Clean Edges** — In facing operations, use negative stock (e.g., -0.125") to push the tool past the inner perimeter and clean up the edge completely.
2. **Handle Multiple Depths in One Pocketing Operation** — Select the bottom curves of all pockets (even at different depths) and set the cut depth based on the **deepest** pocket. The operation will automatically handle varying levels.

3. **Use Bottom Geometry for Pocketing** — Select the bottom curves of pockets rather than top curves for more accurate depth control.
4. **Slotting vs Pocketing** — When the tool diameter exactly matches the slot width, **use the Slotting operation** instead of pocketing (pocketing can fail due to tolerance issues).
5. **Pick Top for Cut Levels** — When geometry is at the bottom but you want to machine from the top (common for outer profiling), use "Pick Top" to set the cut level correctly.
6. **Helical Ramp Entry at Every Level** — Enable this option in pocketing and slotting for safer, more reliable tool entry on every depth pass.
7. **Offset Curves for Step-Down Profiling** — When profiling stepped areas, offset the control curves outward by tool radius + clearance for clean walls.
8. **Single-Stroke Fonts for Engraving** — Use single-stroke (stick) fonts for engraving — they produce much cleaner and faster toolpaths. Contact MecSoft support for recommended font files if needed.
9. **Tool Tip Clearance for Chamfering** — Set a small tool tip clearance (e.g., 0.005") when chamfering to ensure the tool extends slightly below the edge for a complete chamfer.
10. **One Entity Per Hole** — For drilling operations, each hole should be defined by **one entity** (point, arc, or circle). Multiple entities will create multiple drill cycles.
11. **Partial Arcs for Interrupted Holes** — Use partial arcs instead of full circles when holes are interrupted by other features.
12. **Part Files Available** — Download the demonstration part files from the link in the video description to follow along.

Key Takeaways

This webinar is an excellent **practical guide** to 2.5-axis machining for prismatic parts. It shows how to efficiently handle:

- Multiple features in a single setup
- Varying pocket depths in one operation
- Clean facing and profiling with smart stock/curve adjustments
- Text engraving and chamfering
- Best for: Beginners to intermediate users working on plates, brackets, enclosures, and other prismatic components.
- Use 3D models whenever possible for improved setup visualization.
- Establish flat reference surfaces before machining additional features.
- Optimize cut levels and step-down values for efficiency and tool life.
- Verify machining regions carefully before generating toolpaths.

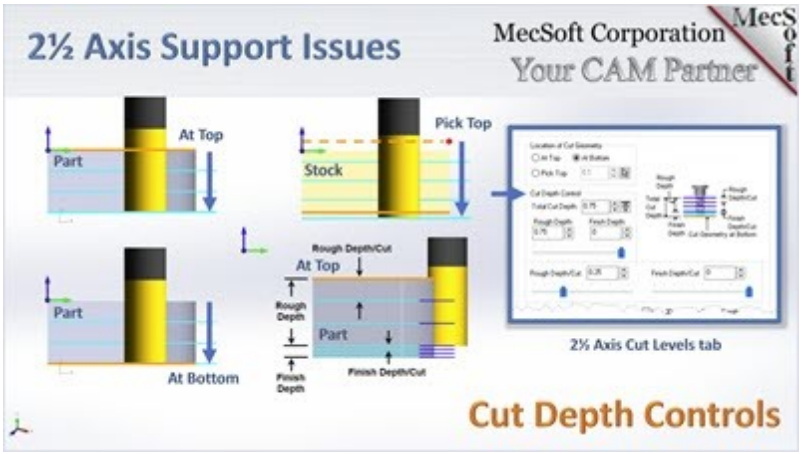
- Use pocketing and profiling strategically for prismatic geometry.
- Validate machining operations using simulation before posting code.

4.21 2.5 Axis Support Issues

Link: https://youtu.be/QmM-VSn4G7w

This AMS Webinar addresses the most common support issues in 2½ Axis machining within RhinoCAM and VisualCAD/CAM. The session combines clear explanations with live demonstrations. It covers geometry handling, stock and work zero setup, cut parameters, tool selection, preferences, posting problems, and real-world machining issues, while also showing how to use support resources effectively.

Source Files: 20.23.04 AMS Webinar 2½ Axis Support Issues.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Housekeeping, presenter introductions, and full topic outline.
4:30	Use of 2D and 3D Geometry	Differences between 2D and 3D geometry; automatic flattening of 3D curves to the highest point.
9:00	Aligning Stock to Part	Stock alignment options (Top/Bottom/Center in Z and X/Y offsets) and why correct positioning matters.
14:00	Establishing Work Zeros	Setting work zeros so they exactly match the machine's physical zero point.

Time	Topic / Section	Description
19:00	Location of Cut Geometry	"At Top", "At Bottom", and "Pick Top" options and their effect on toolpaths.
24:00	Cut Depth Controls	Using total depth, step-downs, and even division for multi-level or varying pockets.
31:00	Cut Start Side in Profiling	Curve direction, start side selection, and the importance of merging curves.
37:00	Tool Geometry and Offsets	Tool types and why ball mills often cause "waterfall effect" edge problems.
43:00	Use of Preferences	Key settings for post processor paths, tooltips, and output dialogue.
49:00	Posting Issues	Common problems with missing posts, file extensions, and save locations.
55:00	Machining Issues	Simulation vs actual results, surface finish, deflection, and setup-related errors.
61:00	Online Support & Resources	Using WebHelp, the presentation PDF, example files, and contacting support effectively.
66:00	Q&A and Closing	Final attendee questions and wrap-up.

Key Tips & Best Practices Extracted

1. **Merge curves before profiling** — Combine separate curves into single entities to prevent gaps, self-intersections, and bad toolpaths.
2. **Use flat end mills** — Avoid ball mills in 2½ Axis operations to prevent the "waterfall effect" that ruins edge quality.
3. **Verify work zero alignment** — The CAM work zero must exactly match the physical machine zero established on the shop floor.
4. **Align stock to part correctly** — Use the alignment tools (Top/Bottom/Center + X/Y offsets) before generating any toolpaths.
5. **Choose the right cut geometry location** — Select "At Bottom", "At Top", or manually pick the reference for accurate depths.

6. **Master cut depth parameters** — Use step-down controls and even division to handle pockets with multiple or irregular bottom levels.
7. **Configure preferences properly** — Set correct post processor paths and enable “Show Output Dialogue” for reliable G-code output.
8. **Save posts in an accessible location** — Keep custom posts on the desktop to avoid “post not found” errors.
9. **Provide complete info to support** — Include G-code, post processor, screen shots, and exact error messages for faster help.
10. **Check machine setup first** — Many sudden issues come from shop-floor changes rather than software problems.

Key Takeaways

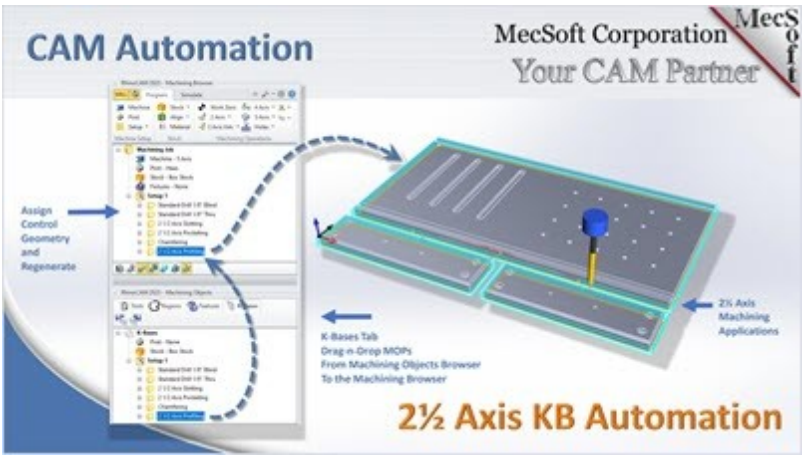
- Most 2½ Axis support issues are caused by geometry preparation, alignment, or parameter settings rather than software bugs.
- Always merge curves and understand that 3D curves are automatically flattened to the highest point.
- Work zero setup is critical and must precisely match the machine’s physical zero.
- Flat end mills are strongly preferred over ball mills for better edge and surface quality.
- Many posting problems are easily fixed by correcting preference settings and post locations.
- Use simulation heavily, but investigate real-world errors by checking machine setup and fixture first.
- Providing detailed information when contacting support leads to much faster and better resolutions.

4.22 CAM Automation

Link: https://www.youtube.com/watch?v=YlFz5KqCMII

This AMS Webinar shows how to automate CAM processes in RhinoCAM and VisualCAD/CAM. In about one hour, it covers templates, tool libraries, default parameters, Knowledge Bases for automatic toolpaths and geometry selection, the RhinoCAM SDK, and Grasshopper integration. The focus is on embedding shop knowledge into reusable systems to cut manual work and boost consistency.

Source Files: 20.23.03 AMS Webinar CAM Automation.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Welcome, speakers, goals, and full agenda overview.
0:30	Template Files	Create reusable templates with fixtures, layers, and geometry for fast, consistent setups.
17:09	Tool Libraries	Machining Objects Browser, tools with embedded feeds/speeds, and drag-and-drop workflows.
17:09	Use of Default Parameters	Save custom operation settings as default Knowledge Bases via CAM Preferences.
22:24	Automatic Toolpath Creation using Knowledge Bases	Save operations to K-Bases; auto-generate toolpaths on new parts by defining stock only.
33:07	Automatic Geometry Selection using Knowledge Bases	Use layers or colors with selection rules for automatic 2.5-axis geometry picking.
43:45	RhinoCAM SDK	Programmatic tool/operation creation, parameter edits, regeneration, and batch processing via Python or C#.

Time	Topic / Section	Description
55:55	Grasshopper in CAM Automation	Visual parametric pipelines linking geometry creation, toolpath generation, and posting using the SDK.
~60:00	Conclusion & Resources	Final tips, SDK docs, learning paths, and support contact.

Key Tips & Best Practices Extracted

1. **Use Template Files:** Save pre-configured fixtures, layers, and geometry as templates for consistent, faster project starts.
2. **Embed Knowledge in Tool Libraries:** Store feeds, speeds, and parameters inside tools so they automatically apply to operations.
3. **Save Defaults as Knowledge Bases:** Customize operations then save them as reusable defaults through CAM Preferences.
4. **Automate 3-Axis with Stock Only:** Define stock and load a Knowledge Base — toolpaths generate automatically with no geometry selection needed.
5. **Standardize Layers or Colors:** Organize CAD geometry consistently so Knowledge Base rules can auto-select it for 2.5-axis operations.
6. **Create Smart Selection Rules:** Build rules in Knowledge Bases using layers, colors, or AND/OR logic for fully automatic geometry assignment.
7. **Drag Specific Operations from K-Bases:** Load only the needed operations from the K-Bases tab instead of entire setups.
8. **Start with SDK Examples:** Use the included Python scripts to learn creating tools, operations, editing parameters, and regenerating toolpaths.
9. **Link Grasshopper to SDK:** Build visual parametric workflows where design changes automatically update geometry and toolpaths.
10. **Standardize for Automation:** Consistent CAD structure and machining strategies make Knowledge Bases and scripts far more reliable and powerful.

Key Takeaways

- CAM automation in RhinoCAM and VisualCAD/CAM works best when shop knowledge is systematically captured in tools, defaults, and Knowledge Bases rather than recreated for every job.

- Template files and well-organized CAD geometry (layers or colors) form the essential foundation that makes higher-level automation reliable and scalable.
- Knowledge Bases can store both operation parameters and intelligent geometry selection rules, enabling true “load part → regenerate” workflows for families of similar parts.
- The RhinoCAM SDK opens the door to fully programmatic control, batch processing, and integration with external systems using Python or C#.
- Grasshopper + RhinoCAM SDK integration allows non-programmers to create powerful parametric design-to-CAM pipelines that update automatically when design parameters change.
- Automation delivers the biggest gains on repetitive or similar parts; investing time in standardization upfront pays off dramatically in reduced programming time and consistency.
- Multiple layers of automation (templates → tool libraries → Knowledge Bases → SDK/Grasshopper) can be combined for maximum efficiency and to support everything from quick previews to full shop-floor automation.

4.23 TURN Basic

Link: https://www.youtube.com/watch?v=k1qCjjxj4T4

This AMS webinar introduces MecSoft’s 2½ Axis TURN module in RhinoCAM TURN and VisualTURN. The presentation walks through the fundamentals of CNC turning workflows, including lathe setup, stock definition, OD and ID turning operations, facing, grooving, threading, drilling, simulation, and post processing. The webinar is designed for users who are new to TURN programming and want to understand how MecSoft’s lathe machining environment works for real-world cylindrical parts and production workflows. Throughout the session, the presenters also share practical turning tips to improve setup accuracy, tool life, machining efficiency, and NC code reliability.

Source Files: 20.22.09 AMS Webinar Intro to 2½ Axis TURN Module.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Module Overview	What the 2½ Axis TURN module can do
5:00	Part Orientation & Definition	Critical setup rules (back face at origin, X-axis alignment)
15:00	Stock & Work Zero Setup	Cylindrical stock definition and correct work zero placement
15:00	Drilling & ID Roughing	Brake chip drilling for ID material removal
30:00	OD Turning (Roughing & Finishing)	OD-forward and OD-backward inserts for full coverage
50:00	ID Turning (Roughing & Finishing)	ID insert orientation and relief angle adjustments
1:10:00	Grooving (Roughing & Finishing)	Groove-specific tools and containment settings
1:30:00	Parting Off	Final cutoff operation with proper tool definition
1:50:00	Simulation, Posting & Best Practices	Full simulation and G-code generation

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Critical Orientation Rule** — The part must be oriented with the back face at the world origin and the rotational axis aligned with the positive X-axis. Incorrect setup causes most turning errors.
2. **Work Zero on the Front Face** — Always set work zero on the front face of the stock (not the back). This ensures correct coordinate output for turning.
3. **Rough ID Before Finishing** — Always clear ID material with drilling or roughing before attempting ID finishing to prevent insert crashes.

4. **Use Two OD Operations** — Most parts require two OD turning operations: one with an OD-forward insert and one with an OD-backward insert to reach all quadrants.
5. **Adjust Relief Angle for ID Finishing** — Rotate the insert relief angle upward when doing ID finishing to prevent material buildup and gouging.
6. **Stock Extension for Drilling** — Extend stock slightly beyond the part so drills have room to exit cleanly.
7. **Define Containment Areas Generously** — Make containment areas larger than the stock to ensure full tool access.
8. **Groove Tools Need Specific Containment** — Set containment exactly to the groove width and use groove-specific roughing direction.
9. **Parting Tool Definition** — Use a straight rectangular parting tool and set the cut position carefully (often near a specific point) for a clean cutoff.
10. **Simulate Everything** — Always run full cut material simulation before posting, especially when switching between OD-forward and OD-backward operations.
11. **Post-Processor Must Support Turning** — Verify your post supports turning cycles (G71, G72, G73, G76, etc.) or request a custom post from MecSoft.
12. **Section Views for Complex Parts** — For non-revolved or complex solids, create a section view and use the resulting curves to define the turning profile.

Key Takeaways

This webinar is the best starting point for anyone new to MecSoft's TURN module. It clearly explains:

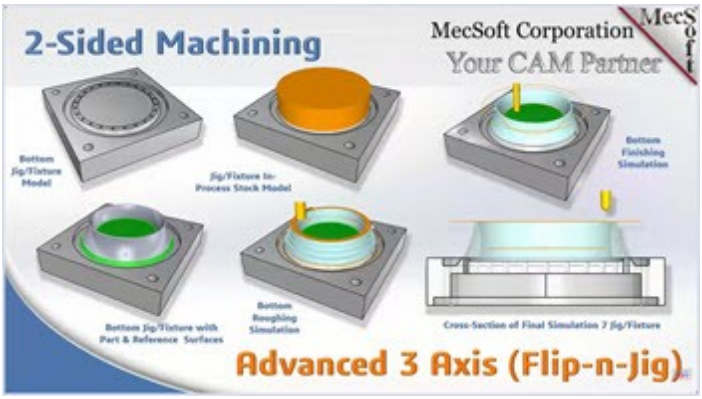
- Best for: Users who have milling experience and want to add turning capability, or shops that do both milling and turning on the same machine.
- The strict orientation and setup rules required for successful turning
- How to handle both OD and ID features with proper tool orientation
- Practical workflows for grooving and parting off
- Prepare clean and fully connected turning profiles before machining.
- Use facing operations to establish reliable reference surfaces.
- Optimize roughing parameters to improve insert life and machining stability.
- Verify tool orientation and clearance motions before posting code.
- Use simulation to validate TURN operations before machining.
- Apply proper OD, ID, grooving, and threading workflows for cylindrical parts.

4.24 Flip Machining

Link: https://www.youtube.com/watch?v=r9wWhJVWeHE

This AMS webinar focuses on 2-sided “flip” machining workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how parts can be machined from multiple sides by flipping stock while maintaining accurate alignment between setups. Topics include setup creation, machining coordinate systems (MCS), alignment strategies, stock preparation, work zero management, roughing and finishing operations, simulation, and post processing. Throughout the webinar, the presenters also share practical shop-floor tips for improving setup accuracy, reducing alignment errors, and machining complex two-sided parts efficiently using MecSoft CAM workflows.

Source Files: 20.22.08 AMS Webinar 2-Sided Flip Machining.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Why Flip Machining is Needed	Identifying undercuts and parts that require two sides
15:00	Standard vs Professional Configuration Differences	One setup per file vs multiple setups in one file
30:00	Strategy 1: Bridge and Flip (Basic 2.5-Axis)	Using tabs/bridges to keep the part attached during flip
50:00	Strategy 2: Box and Flip (Basic 3-Axis)	Modeling tabs/supports and using box stock

Time	Topic / Section	Description
1:10:00	Strategy 3: Flip and Jig (Advanced 3-Axis)	Using custom fixtures/jigs for precise registration after flipping
1:30:00	Live Demo: Standard vs Professional Workflows	Side-by-side comparison of file management and alignment
1:45:00	Q&A, Post-Processing & Best Practices	Post settings, work zero consistency, and final tips

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Alignment is Everything** — Always center the part and stock in X, Y, and Z. Use the **same work zero location** (e.g., bottom-left corner) before and after flipping to maintain registration.
2. **Standard Configuration = Two Files** — In Standard, you must save the part as two separate files (one for each side) and manually flip/realign in the second file. This increases the risk of misalignment errors.
3. **Professional Configuration = One File, Multiple Setups** — Add a second setup, rotate the part 180° about the X-axis, and the software maintains alignment between part and stock automatically. Much safer and faster.
4. **Bridge and Flip (Simplest Method)** — Add tabs or bridges to the toolpath on the first side so the part stays attached to the stock. After flipping, machine the second side to cut the part free.
5. **Box and Flip (More Secure)** — Model physical tabs, cylinders, or blocks into the part geometry to hold it to the stock. This works well for 3-axis parts with internal features.
6. **Flip and Jig (Most Precise)** — Machine a custom fixture/jig alongside the part. After flipping, place the part into the jig for perfect registration. Ideal for parts with critical edges or flanges.
7. **Hide Stock Geometry After Definition** — In 3-axis operations, hide the stock model so toolpaths can extend beyond its boundaries when needed.
8. **Use Consistent Flip Axis** — Always flip about the same axis (usually X) and match your machine's actual rotation direction.
9. **Tabs Must Extend Beyond Containment** — When using tabs, make sure they extend past your containment curves so the part doesn't separate prematurely.

10. **Post-Processor Considerations** — Ensure your post outputs setup-specific information (e.g., work plane changes or comments between setups). You can insert pause commands between setups using the Post Process Editor.
11. **Vacuum Tables Still Need Tabs** — Even with vacuum hold-down, you usually need bridges or tabs to keep the part stable while flipping and during the second-side machining.
12. **Test Alignment with Simulation** — Always simulate both sides fully before posting to verify that features on the second side align correctly with the first side.

Key Takeaways

This webinar is excellent for anyone who regularly machines parts with features on both sides. The three strategies scale from simple to advanced:

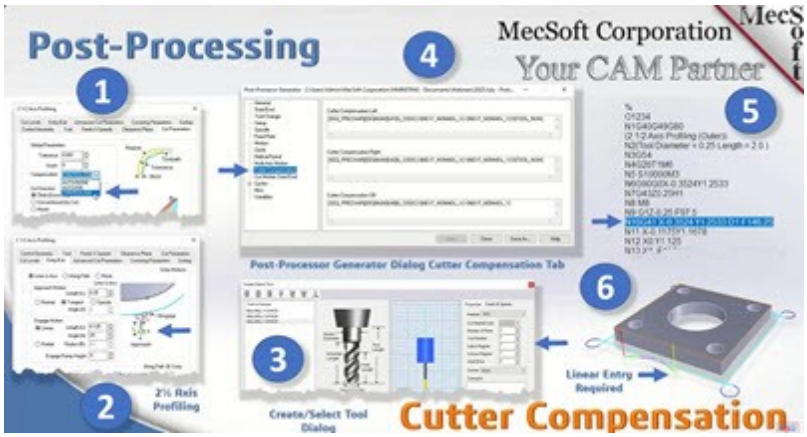
- Bridge and Flip — Quickest for basic 2.5-axis work
- Box and Flip — More robust for 3-axis parts
- Flip and Jig — Most precise for complex or high-accuracy parts
- Plan alignment and setup strategy before creating toolpaths.
- Use alignment pins and registration holes for accurate stock flipping.
- Maintain consistent work zero locations across setups.
- Use control geometry and tabs to preserve stock stability.
- Simulate both machining setups before cutting material.
- Verify flipped orientation carefully before posting code.
- Biggest advantage highlighted: The Professional configuration dramatically simplifies 2-sided machining by allowing multiple setups in a single file with automatic alignment.

4.25 Post Advanced

Link: https://www.youtube.com/watch?v=SdCoAhlX-zc

This AMS webinar focuses on post-processing workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how post processors convert CAM toolpaths into machine-specific NC code and how users can configure post-processing settings for different CNC machines and controllers. Topics include selecting post processors, posting operations, understanding NC output, editing post settings, managing machine compatibility, troubleshooting output issues, and verifying generated code. Throughout the webinar, the presenters also share practical tips for improving reliability, reducing posting errors, and streamlining CNC programming workflows.

Source Files: Unavailable



1 of 15 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:20	Introduction & Agenda	Why post-processing matters and what will be covered
2:00	Common Posting Issues & Troubleshooting	Lost posts, file extensions, G-code viewer, upgrade safety
13:30	New Features	G-Code flag icon and auto-post option
22:00	Editing Post Processors	How to access and modify posts
25:00	Tool Change Points	Safe retract locations and macro setup
26:00	Cutter Compensation (G41/G42/G40)	When and how to use it correctly
33:00	Arc/Circle Formats	I,J vs R output, plane support, tolerance settings
41:00	Drill Cycles as Linear Motions	Converting canned cycles for older machines
42:00	Programmable Post Processors	Python-based custom posts for advanced needs
1:05:00	Demo & Wrap-Up	Live examples and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Protect Custom Posts During Upgrades** — Store your custom post processors in a separate folder (not the default installation folder). Use the "Folder to Load Post Processor Files From" setting in CAM Preferences to point to it. This prevents them from being overwritten during service pack updates.
2. **Tool Change Points for Safety** — Define a safe tool change/retract point (X, Y, Z) in the Machine Tool Setup dialog. Enable the corresponding macros in the post processor's Tool Change tab so the tool always retracts to a known safe position between operations.
3. **Cutter Compensation Rules:**
    - Only enable on the final pass of profiling or the cleanup pass of pocketing.
    - Must have a linear entry motion (no ramps or arcs during engagement).
    - Post must include G41 (left), G42 (right), and G40 (cancel) macros.
    - Always start programs with G40 to ensure compensation is off.
4. **Arc Fitting Best Practice** — Set arc fitting tolerance to approximately 2× your global tolerance. This gives good arc output without creating too many small segments.
5. **Match Arc Output to Your Machine** — Check your machine manual for preferred arc format (I,J vs R vs I,J,K) and supported planes (G17/G18/G19). Incorrect formats are a common source of controller alarms.
6. **Convert Drill Cycles for Older Machines** — If your machine doesn't support canned cycles (G81–G89), enable "Always Output Drill Cycle Motions as Linear Motion" in CAM Preferences.
7. **Programmable Posts** — Use Python-based programmable posts when you need advanced logic (e.g., custom 5-axis indexing, special output formats, or machine-specific variables like U,V,W). MecSoft offers paid custom post development for complex requirements.
8. **Test Before Full Posting** — Always post and test a single operation first before posting an entire job. This makes troubleshooting much faster.
9. **Use the G-Code Flag** — The small G-Code icon on the MOP folder now shows you at a glance which operations haven't been posted or regenerated yet.
10. **Central Post Storage** — Keep all your custom posts in one shared/network folder so the entire team uses the same versions and nothing gets lost.
11. **Linear Entry for Cutter Compensation** — When using cutter compensation, force a straight-line entry move. Ramps or arcs during engagement will cause the controller to ignore or error on the compensation command.

Key Takeaways

This webinar is one of the best resources available for understanding and mastering post-processing in MecSoft CAM. It covers both everyday troubleshooting and advanced customization options.

Biggest lessons:

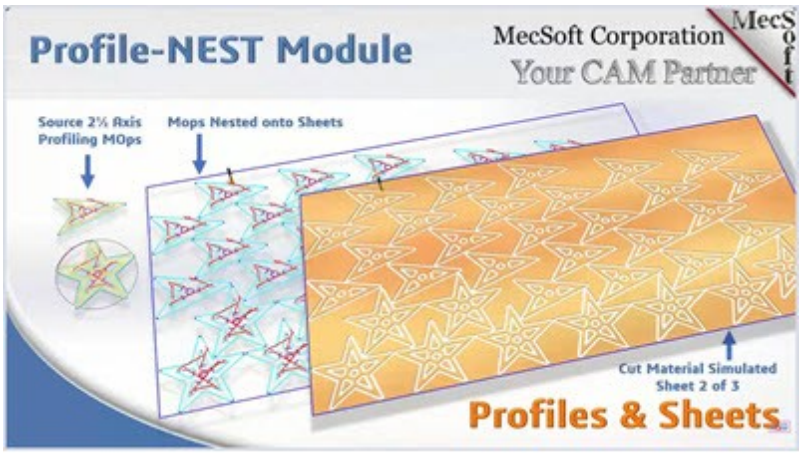
- Most post-processing problems come from setup and file management rather than the post itself.
- Cutter compensation and arc output are two of the most common sources of G-code errors.
- The new programmable posts (Python-based) open up powerful customization possibilities.
- Always match the post processor to the CNC controller.
- Review NC code before transferring programs to the machine.
- Standardize output formatting across jobs and machines.
- Use simple test programs when troubleshooting post issues.
- Organize post processors carefully in multi-machine environments.
- Validate machine motion with simulation before production.

4.26 Profile NEST

Link: https://www.youtube.com/watch?v=RIuStZ-LxW4

This AMS webinar introduces MecSoft's Profile-NEST module for RhinoCAM and VisualCAD/CAM. The presentation focuses on sheet nesting workflows used in CNC router and flat-sheet manufacturing applications. Topics include staging parts, defining sheet stock, creating profile operations, nesting parameters, tool setup, feeds and speeds, entry and exit motions, cut parameters, simulation, and post processing. Throughout the webinar, the presenters also share practical nesting and production tips for improving material utilization, reducing waste, and streamlining CNC cutting workflows.

Source Files: 20.22.06 AMS Webinar Introduction to Profile NEST.zip



1 of 11 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Module Overview	Profile NEST vs Standard Nesting Module
3:00	Geometry & Tooling Supported	2D, wireframe, 3D poly surfaces; supported tools
7:00	Setting Up 2.5-Axis Profiling Operations	Cut parameters, nesting-specific settings (quantity, priority, grain)
11:00	Sheet Definition	Defining sheets by dimensions or curves
12:00	Nesting Parameters	Rectangular vs True Shape, spacing, orientation, tagging
16:00	**Live Demo: Executing & Editing a Nest**	Setting up operations, running the nest, simulation, editing
42:00	Reporting & Post-Processing	Generating reports, posting nested sheets, Knowledge Bases
End	Tips, Best Practices & Q&A	Final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Profile NEST Nests Toolpaths, Not Just Geometry** — This is the biggest advantage. It accounts for the full toolpath (including entry/exit moves), making nesting safer and more accurate than standard geometry nesting.
2. **Use "Long Path" or Ramp Entry/Exit** — When setting up profiling operations for nesting, use long-path or ramp entries. This ensures the nesting algorithm sees the full extent of the toolpath and avoids collisions.
3. **Priority & Fixed Orientation** — Use **Priority** to control which parts are placed first (higher priority parts go bottom-left). Use **Fixed Orientation** when parts must stay in a specific rotation (e.g., for grain direction or aesthetics).
4. **Break Complex Parts into Separate Operations** — For parts with floating segments or complex shapes (e.g., a star with disconnected pieces), create separate profiling operations. This gives you more control during editing and re-nesting.
5. **True Shape vs Rectangular Nesting** — Use **True Shape** for tighter packing of irregular parts. Use **Rectangular** when you want simpler, faster calculations.
6. **Part-to-Part Spacing = Toolpath Center-line Distance** — Set this spacing based on your tool diameter + desired clearance between finished parts.
7. **Knowledge Bases for Repeatability** — Save your profiling operation settings (tolerances, step-over, entry/exit, etc.) as a Knowledge Base so you can quickly apply the same parameters to new parts.
8. **Edit Individual Operations After Nesting** — You can edit specific operations (e.g., change entry/exit on one part) without re-running the entire nest. This is very powerful for fine-tuning.
9. **Reports Include Estimated Cut Times** — The module generates reports showing quantity per sheet and estimated machining time based on your feed rates — useful for quoting and scheduling.
10. **Post Using Standard Mill Posts** — Profile NEST uses the same post-processors as the Mill module, so it supports flame cutters, waterjets, routers, etc.
11. **Separate Sheets by Thickness** — If you have parts of different material thicknesses, create separate sheets rather than trying to nest them together.

Key Takeaways

This webinar clearly explains why the Profile NEST Module is a valuable AMS-exclusive tool:

- Best for: Shops doing production nesting of 2.5-axis profiled parts (signs, brackets, enclosures, decorative panels, etc.) who want better material utilization and fewer collisions.
- It gives you more control and safety by nesting actual toolpaths instead of just outlines.
- It integrates seamlessly with 2.5-axis profiling operations.

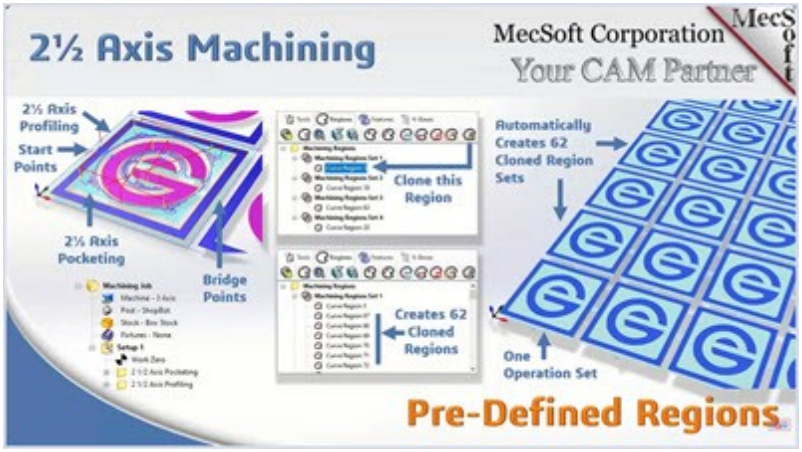
- Editing and re-nesting is flexible and efficient.
- Create separate profile operations for better nesting control.
- Standardize sheet sizes to improve production efficiency.
- Use alternate cut-side logic for interior cutouts.
- Apply ramp entry motions to reduce tool stress and improve cut quality.
- Optimize arc fitting and cornering parameters for smoother machine motion.
- Verify nested layouts with simulation before machining.

4.27 2.5 Axis Bootcamp

Link: https://www.youtube.com/watch?v=4OfStmaSCOc

This AMS webinar focuses on CAMJam Bootcamp training for 2½ Axis machining workflows in RhinoCAM and VisualCAD/CAM. The presentation is designed as a practical training session that walks users through core 2½ Axis machining concepts used in real-world CNC programming. Topics include machining setup, stock definition, tool selection, facing, profiling, pocketing, drilling, cut levels, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining efficiency, reducing setup errors, and building reliable CNC machining workflows for prismatic parts and production machining environments.

Source Files: 20.22.05 AMS Webinar CAMJam Bootcamp for 2½ Axis Machining.zip



1 of 16 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Bootcamp Overview	Goals of the 2.5-axis Bootcamp

Time	Topic / Section	Description
3:09	2½ Axis Explanation	How 2.5-axis differs from full 3-axis (Z fixed at levels)
5:03	Cutting Tools & Contact Areas	Flat, ball, V, and chamfer mills — when to use each
6:46	Control Geometry	Lines, curves, 2D/3D sources, closed vs open curves, avoid regions
22:47	Cut Levels & Depth Control	Top/Bottom geometry location, Pick Top, roughing vs finishing depths
27:13	Entries and Exits	Linear, arc, ramp, helical options
30:10	Live Demonstration Begins	Practical examples on a complex part
30:40	Facing	Outer perimeter and open pocket facing
35:15	Step Pocketing (with Islands)	Multi-level pocketing with “Clear Island Tops”
41:17	Profiling (Finishing)	Pocket edge finishing with ramp entry
44:27	Slotting (Trichoidal)	Using “Treat Curve as Spine” with trichoidal cutting
46:16	Chamfering	V-mill chamfering with tool tip clearance
48:37	Whole Pocketing (Through Holes)	Helical entry + circular cleanup at each level
50:26	Outer Profiling + Sharp Corners	Final outer profile and sharp corner handling
58:05	Wrap-Up & Q&A	Final tips and closing

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Avoid Ball Mills in 2.5-Axis** — Ball mills create a “waterfall effect” that can degrade the top edge of pockets due to tolerance stack-up. Use flat mills unless you specifically need a fillet at the bottom.
2. **Use “Pick Top” for Flexible Depth Control** — When control geometry is at the bottom but you want to start cutting from above (e.g., spoil board leveling), use the Pick Top feature to define the starting Z height.
3. **Closed Curves for Pocketing/Facing** — Pocketing and facing require closed control geometry. The software can internally close open pockets with a virtual line for boundary calculation.
4. **“Clear Island Tops” for Step Pocketing** — Enable this option so island tops are machined at their correct Z-level while the main pocket is roughed in multiple steps.
5. **“Treat Curve as Spine” for Slotting** — When the curve represents the center line of a slot (instead of the edge), use this option. The software will automatically apply a trichoidal (high-speed, constant engagement) pattern if the tool is smaller than the slot width.
6. **Tool Tip Clearance for Chamfering** — Set a small tool tip clearance (e.g., 0.010") when chamfering so the tip cleans below the edge for a complete chamfer without leaving a witness line.
7. **Helical Entry + Arc Fitting** — Use helical entry for pocketing and enable arc fitting (set tolerance ~2× global tolerance) for smoother, faster toolpaths.
8. **Predefined Regions + Cloning** — Create named regions in the Machining Browser and use the Cloning feature to automatically find and populate similar geometry across multiple instances or arrays.
9. **Curve Direction Matters** — The direction and start point of a curve determine the cutting side (left/right rule). Merge curves into single entities for cleaner selection and validation.
10. **Roughing vs Finishing Depths** — Clearly separate roughing and finishing depths. You can divide total depth into multiple roughing levels + one finishing level.
11. **Avoid Regions Must Bisect** — When using Avoid Regions, they must intersect (bisect) the control geometry, not just overlap it.
12. **Sharp Corners Option** — In profiling, enable “Sharp Corners” to maintain crisp corners instead of rolling around them with the tool radius.

Key Takeaways

This CAMJam Bootcamp delivers excellent value by combining theory with live demonstration on a realistic complex part. It’s one of the best resources for understanding the nuances of 2.5-axis machining beyond basic pocketing and profiling.

Biggest lessons:

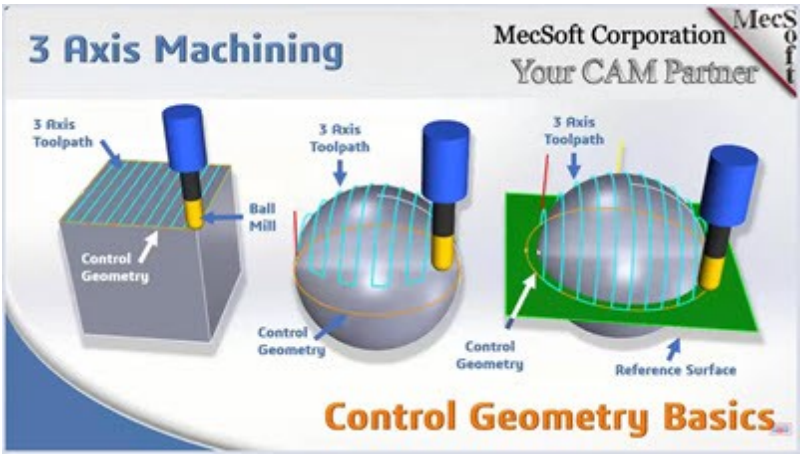
- Proper control geometry setup and curve management is 80% of successful 2.5-axis work.
- Small settings like “Clear Island Tops,” “Treat Curve as Spine,” and tool tip clearance make a big difference in finish quality and reliability.
- The software is very flexible — many operations can be adapted creatively (e.g., using profiling for finishing or slotting with trichoidal paths).
- Prepare clean geometry before creating machining operations.
- Build reusable and standardized tool libraries.
- Optimize pocketing parameters for improved machining efficiency.
- Use hole recognition workflows to simplify drilling operations.
- Understand cut levels and step-down settings for better machining control.
- Verify all toolpaths using simulation before posting code.

4.28 2 & 3 Axis Machining

Link: https://www.youtube.com/watch?v=Zsnqea2HY14

This AMS webinar focuses on helping users understand the core differences and workflows between 2 Axis, 2½ Axis, and full 3 Axis machining in RhinoCAM and VisualCAD/CAM. The presentation explains how each machining strategy works, what types of geometry each method supports, and how users can select the appropriate machining approach for real-world CNC applications. Topics include setup preparation, stock definition, profiling, pocketing, drilling, 3-axis surfacing, roughing, finishing, simulation, and post processing. Throughout the webinar, the presenters also share practical programming tips for improving machining efficiency, surface finish quality, and workflow organization.

Source Files: 20.22.03 AMS Webinar Understanding 2 & 3 Axis Machining.zip



1 of 18 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Welcome, series overview, and agenda
3:37	General CAM Workflow	Standard workflow: part/stock → setup → operations → simulate
6:05	Machine & Setup Configuration	Setup One, Work Zero, Standard vs Expert limitations
8:30	Geometry Types	Wireframe, solids, poly surfaces, meshes
10:15	2.5-Axis vs 3-Axis Definitions	Core differences in tool movement and control
12:08	2.5-Axis Operations Explained	Facing, Pocketing, Profiling, Engraving
15:56	Cut Levels & Tool Interaction	Tool types, “waterfall effect,” stock allowances
20:53	3-Axis Concepts & Offset Curves	Parallel finishing, Z containment, Clear Flats
29:44	Post-Processing & Resources	600+ posts, online help, PDF tutorials
37:30	Demo: 2.5-Axis Prismatic Part	Facing, pocketing (flat areas), profiling with arcs
48:55	Demo: 3-Axis Curved Part (Hair Dryer Core)	Horizontal roughing, parallel finishing, fillet profiling
1:01:54	Conclusion & Q&A	Support contact and final recap

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Always Simulate Every Operation** — This is the #1 best practice. Simulate after every change to catch collisions, verify material removal, and prevent tool damage.
2. **Work Zero vs Setup** — In Standard/Expert configurations, the Setup is locked to the World Origin. Use Work Zero to freely move the machining origin (top of stock, bottom of part, etc.).
3. **Merge Curves for Clean Toolpaths** — Always merge multiple curves into one continuous curve before using them for profiling or containment. This ensures consistent start points and direction.

4. **Avoid Ball Mills for 2.5-Axis Pocketing** — Ball mills create a "waterfall effect" — the tool re-cuts the same vertical edge at every depth level, causing poor finish and tolerance buildup. Use flat end mills for pocketing and ball mills only for contoured finishing.
5. **Use Offset Curves in 3-Axis Finishing** — When finishing with a ball mill or corner radius mill near a parting plane or fillet, create an offset curve (larger than tool radius) so the tool can reach the bottom without gouging.
6. **"Clear Flats" Option** — Enable this in horizontal roughing/finishing when you want the tool to fully machine flat areas (especially useful for mold work).
7. **Curve Direction Matters** — The direction and start point of a curve determines cutting side (Inside/Outside for closed curves). Always check direction before generating toolpaths.
8. **Z Containment for Safety** — Use Z containment to limit machining to specific height ranges, especially on complex 3D parts.
9. **Clone Operations for Efficiency** — After creating one good operation (e.g., a through-hole), clone it and simply reassign new geometry instead of recreating from scratch.
10. **Part Files & Resources Available** — Download the demo parts and step-by-step PDF tutorials from the Online Help > Resource Guide > Mill Module.

Key Takeaways

This webinar is an excellent foundational training that clearly explains the "why" behind 2.5-axis and 3-axis strategies before showing the "how" in live demos.

Biggest lessons:

- The session perfectly bridges theory and real-world application, making it ideal for new users and those transitioning from 2D to 3D machining.
- 2.5-axis = tool fixed at Z levels, controlled by curves
- 3-axis = tool free to move in X/Y/Z, controlled by the 3D model
- Proper tool selection and offset curves are critical for clean results
- Simulation is non-negotiable
- Use the simplest machining strategy capable of producing the required geometry.
- Use 3D models whenever possible for improved setup visualization.
- Optimize cut levels and step-down settings for better machining efficiency.
- Balance surface finish quality against machining cycle time.
- Match machining strategies to the geometry being manufactured.
- Use simulation to validate setups and toolpaths before machining.

4.29 2.5 Axis Special

Link: https://www.youtube.com/watch?v=QpIfKWzKz8Y

This AMS webinar focuses on advanced and specialized topics related to 2½ Axis machining in RhinoCAM and VisualCAD/CAM. The presentation explores practical machining workflows used for complex prismatic parts, including advanced cut-level management, multi-depth machining, profiling strategies, pocketing optimization, drilling workflows, machining regions, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining efficiency, reducing setup errors, and optimizing 2½ Axis workflows for real-world manufacturing applications.

Source Files: 20.21.11 AMS Webinar Special Topics in 2½ Machining.zip

1 of 9 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction and Housekeeping	Overview of the five special topics
3:15	Aggregate Head Machining Overview	Concept and machine definition setup
9:15	Demo: Aggregate Head Machining	Setup, multiple coordinate system rotations, post-processing with manual rotation prompts
5:00	Disk Machining (Rotary Saws) Overview	Defining saw blades as face mills

Time	Topic / Section	Description
27:15	Demo: Disk/Saw Machining	Profiling and T-slotting with negative stock for saw cuts
6:00	Drill Banks (Multi-Drill Heads) Overview	Production drilling with binary encoding
32:25	Demo: Multi-Drill Head Programming	Creating custom tools with binary codes and programmable post output
6:40	Knife Cutting Overview	Drag knives vs Rotary (swivel) knives
47:55	Demo: Knife Cutting	Swivel depth, corner angle limits, and curve orientation
8:00	Machine Control Operations Overview	Inserting custom G-code and Op Stops (M00)
55:25	Demo: Machine Control Operations	Creating Operation Stops for manual tasks
End	Q&A and Wrap-Up	Final questions and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Aggregate Heads = 5-Axis Simulation on 3-Axis Machines** — Define a 5-axis head-to-head machine in the software even on a 3-axis mill. Create multiple setups by rotating the coordinate system (Z-axis) to face different sides. The post outputs rotation angles as comments (e.g., M00) so the operator manually rotates the aggregate head between setups.
2. **Gauge Length is Critical** — When setting up aggregate heads, always enter the correct **gauge length** in the machine tool definition. This ensures accurate coordinate transformations after rotation.
3. **Define Saws as Face Mills** — For disk/saw machining, create a **face mill** tool where:
    - Cutter diameter = saw blade diameter
    - Flute length = saw blade thickness Use **negative stock** in profiling operations to control cut depth.

4. **Binary Encoding for Drill Banks** — Assign each drill in a multi-drill head a binary value (1, 2, 4, 8, 16…). Create custom tools with combined binary numbers (e.g., Tool 7 = Drills 1+2+3). The programmable post-processor (Python) decodes the tool number and outputs the correct on/off commands for each drill.
5. **Swivel Depth for Drag Knives** — Set a small **swivel depth** so the knife can rotate freely at corners without lifting completely out of the material.
6. **Corner Angle Limit for Rotary Knives** — Use the **corner angle limit** setting to automatically pick up the knife and rotate it at sharp corners (prevents tearing or incomplete cuts).
7. **Curve Orientation Matters for Knives** — Ensure curves are oriented correctly for unidirectional knives. The software assumes bi-directional cutting by default in older versions.
8. **Machine Control Cycles for Op Stops** — Use **Machine Control Cycles** to insert M00 (program stop) commands with custom messages (e.g., “Rotate aggregate head to 90°”). This is perfect for machines without automatic tool changers or when manual intervention is required.
9. **Programmable Posts Are Powerful** — Many of these special operations (especially drill banks and custom machine commands) rely on **Python-based programmable post-processors** for full functionality.
10. **Test One Feature at a Time** — When implementing any of these special topics, start with a simple test part before applying to production jobs.

Key Takeaways

This webinar is a **goldmine for advanced 2.5-axis users** who want to push beyond standard pocketing and profiling. The five topics covered can dramatically increase productivity on the right machines:

- Best for: Shops with CNC routers, panel saws, or machines equipped with aggregate heads, multi-drill blocks, or knife tooling.
- Aggregate heads for multi-sided work on 3-axis mills
- Saw/disk machining for fast straight cuts
- Multi-drill banks for high-volume hole production
- Knife cutting for signage, composites, and soft materials
- Machine Control Operations for custom automation
- Organize machining operations based on setup strategy.
- Optimize multi-depth step-down values for improved tool life.
- Verify machining regions carefully before generating toolpaths.
- Optimize linking moves to reduce unnecessary machine motion.

- Use advanced pocketing and profiling strategies for complex prismatic geometry.
- Validate machining workflows using simulation before posting code.

4.30 System Performance

Link: https://www.youtube.com/watch?v=Wc64AIvv0GA

This AMS webinar focuses on improving CAM system performance in RhinoCAM and VisualCAD/CAM. The presentation explains how users can optimize software settings, hardware resources, toolpath workflows, simulation performance, graphics settings, and machining strategies to achieve faster and more reliable CAM programming. Topics include toolpath regeneration, display optimization, simulation tuning, file management, hardware considerations, and workflow organization. Throughout the webinar, the presenters also share practical productivity tips for reducing lag, improving toolpath calculation speed, and creating more efficient CAM programming environments.

Source Files: 20.21.10 AMS Webinar Fine-Tune your CAM System Performance.zip

1 of 14 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of performance tuning topics
4:07	Hardware Performance	Base vs recommended requirements, CPU, RAM, SSD, graphics cards
8:17	System Tips	Nightly shutdown, single sessions, auto-save, process priority

Time	Topic / Section	Description
11:48	Demo: Checking System Info	Using Windows System Information and Device Manager
18:13	Geometry Performance	2D curves vs NURBS vs meshes – which is fastest
21:00	Demo: Mesh Density Impact	700k triangle mesh vs 50k triangle mesh (35s vs 10-15s)
27:27	Tool Path Generation Parameters	Tolerance, step-over, cut patterns, multi-threading
35:35	Demo: Tolerance Impact	Tightening tolerance from 0.005" to 0.01" (10× slower)
40:47	Demo: Multi-Threading	30 operations with multi-threading on vs off
42:06	Simulation Performance	Voxel vs Polygonal models, display interval, Simulate Until
45:50	Demo: Voxel Simulation Speed	Dramatic speed difference between models
56:00	Conclusion & Final Tips	Summary and closing recommendations

Key Tips & Best Practices Extracted

Here are the most valuable performance tips from the webinar:

Hardware & System Level

1. **Nightly Shutdown** — Shut down your computer every night. This releases locked memory caused by system library leaks and keeps long-term performance stable.
2. **Single Session Only** — Run only one instance of RhinoCAM / VisualCAD/CAM at a time. Multiple sessions steal resources from each other.
3. **Set Process Priority to Real-Time** — In Task Manager → Details tab, right-click the CAM process and set priority to Real-time during heavy calculations or simulations.
4. **Enable Auto-Save Every 5 Minutes** — Prevents data loss during long toolpath generations.
5. **Keep Graphics Drivers Updated** — Use the latest certified drivers for your GPU (especially NVIDIA Quadro or RTX series).

Geometry & Toolpath Generation

6. **2D Curves Are Fastest** — Use 2D curves whenever possible. NURBS surfaces are acceptable. 3D meshes are slowest — avoid them when you can.
7. **Reduce Mesh Density** — If you must use meshes, request lower triangle counts from clients or use mesh reduction tools. A 50,000-triangle mesh can generate toolpaths 2–3× faster than a 700,000-triangle mesh.
8. **Loosen Tolerance When Possible** — Tight tolerances (e.g., 0.001") dramatically increase calculation time. Use 0.005"–0.01" for roughing and only tighten for final finishing.
9. **Use Fast Cut Patterns** — Offset and Linear patterns are fastest. Avoid Spiral and Radial patterns unless necessary (they are much slower).
10. **Enable Multi-Threading** — Go to Machining Preferences → General and check "Always generate toolpaths in multiple threads". This can cut generation time by 50% or more on multi-core CPUs.
11. **Hide Unnecessary Layers** — Turn off layers containing geometry not used in the current operation before generating toolpaths.

Simulation Performance

12. **Use Voxel (Box) Model for 2.5-Axis & 3-Axis** — Voxel simulation is over 300% faster than Polygonal simulation for most operations (limited to single Z-axis setups).
13. **Increase Maximum Display Interval** — Change from 100 to 500–1000 moves so the screen refreshes less often during simulation.
14. **Use "Simulate Until"** — Right-click any operation → Simulate Until to jump directly to the final result without simulating every intermediate step.
15. **Use Standard or Medium Accuracy** — Fine accuracy settings slow simulation significantly. Use them only when needed.
16. **Avoid Machine Tool Simulation unless required** — It adds ~15% to simulation time.

Key Takeaways

This webinar is one of the best practical performance guides available for MecSoft CAM users. Many of the tips are still highly relevant in 2026 versions.

Biggest performance wins:

- The session proves that smart settings often outperform hardware upgrades.
- Multi-threading + reasonable tolerances = massive time savings on toolpath generation.
- Voxel simulation + higher display interval = dramatically faster verification.
- Mesh reduction + hiding unused geometry = quick wins on complex parts.

- Simple system habits (single session, nightly reboot, Real-Time priority) = free speed.
- Prioritize CPU performance for faster toolpath calculations.
- Simplify geometry to improve regeneration and simulation speed.
- Optimize CAM Preferences and default settings before programming.
- Use efficient machining strategies to reduce calculation time.
- Organize CAM projects carefully for smoother workflows.
- Keep CAM workstations optimized and free from unnecessary background tasks.

4.31 Mold Machining

Link: https://www.youtube.com/watch?v=OlkoFDNdKKQ

This AMS webinar focuses on programming CNC toolpaths for injection mold machining using RhinoCAM and VisualCAD/CAM. The presentation walks through practical workflows for machining mold cavities, cores, inserts, and contoured mold surfaces using advanced 3-axis and multi-axis machining strategies. Topics include mold geometry preparation, stock setup, roughing methods, finishing operations, cutter selection, tool containment, simulation, and post processing. Throughout the webinar, the presenters also share practical mold-making tips for improving surface finish quality, reducing machining time, and creating reliable CNC workflows for injection mold manufacturing.

Source Files: 20.21.09 AMS Webinar Programming toolpaths for injection mold machining.zip

1 of 19 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of injection mold machining challenges

Time	Topic / Section	Description
4:30	Part Overview & Geometry Analysis	Mold core features, parting plane, draft angles, fillets
10:00	Recommended Tools & Stock Setup	Flat end mills vs ball mills, stock definition, work zero
15:00	Horizontal Roughing	Z-level roughing with step minimization and masking surfaces
28:00	Parallel Finishing (Primary Finishing Pass)	X & Y direction passes, step-over, tolerance settings
38:00	Horizontal Finishing (Sidewalls & Vertical Areas)	When and why to use it for mold walls
45:00	Parting Plane Handling & Guide Surfaces	Critical technique for maintaining parting line integrity
52:00	Pencil Trace / Valley Machining	Cleaning sharp corners and fillets (Professional feature)
58:00	Simulation, Post-Processing & Best Practices	Full job simulation and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Never Machine Directly to the Parting Plane with a Flat End Mill** — Always leave 0.010"–0.025" stock on the parting plane during roughing and early finishing. Finish it later with a ball mill or corner radius mill to maintain tangent contact and protect the parting line.
2. **Use Guide Surfaces for Parting Line Integrity** — Add temporary guide surfaces (extruded or revolved) that extend past the parting plane. This forces the tool to cut **past** the parting line in X/Y while staying tangent in Z, preventing the tool from riding along or degrading the parting line edge.
3. **Run Parallel Finishing in Two Directions (0° + 90°)** — This dramatically improves surface finish on contoured mold surfaces and reduces the need for hand polishing.

4. **Step Minimization is Extremely Effective for Molds** — Combine roughing + re-roughing into a single operation by enabling step minimization (e.g., 25–50% of tool diameter). This saves significant time on deep cavities.
5. **Masking Surfaces Protect Critical Areas** — Create masking surfaces to prevent the tool from entering pockets, holes, or areas you want to leave stock on during roughing.
6. **Pencil Trace / Valley Machining for Sharp Corners** — Use this Professional feature to clean up sharp internal corners and fillets that standard parallel or horizontal finishing cannot reach cleanly.
7. **Ball Mills for Finishing Only** — Use flat end mills for all roughing (faster calculation and material removal). Switch to ball mills only for final finishing passes.
8. **Clear Flats Option** — Enable “Clear Flats” in horizontal finishing operations to ensure the tool fully machines flat areas instead of lifting prematurely.
9. **Offset Curves for Safe Finishing Near Edges** — When finishing near parting lines or draft walls, offset your containment curves outward by tool radius + clearance to keep the tool from gouging or rolling off the edge.
10. **Simulation is Non-Negotiable** — Always run full cut material simulation with fine accuracy before posting, especially when using guide surfaces or multiple finishing directions.
11. **Stock Allowance Strategy** — Rough with 0.030"–0.050" stock, semi-finish with 0.010"–0.015", and finish with 0.000"–0.005" for mold-quality surfaces.

Key Takeaways

This webinar is an excellent **practical guide** for anyone programming injection molds. The biggest emphasis is on **protecting the parting line** — one of the most critical (and mistake-prone) areas in mold machining.

Core workflow highlighted:

- The techniques shown here remain highly relevant and are still used in current MecSoft CAM versions.
- Rough with flat end mills + step minimization
- Parallel finish in two directions
- Use guide surfaces + proper stock on parting plane
- Pencil trace / horizontal finishing for final cleanup
- Organize mold geometry carefully before creating toolpaths.
- Optimize roughing workflows to reduce finishing cycle time.
- Use rest machining to clean up difficult mold features efficiently.

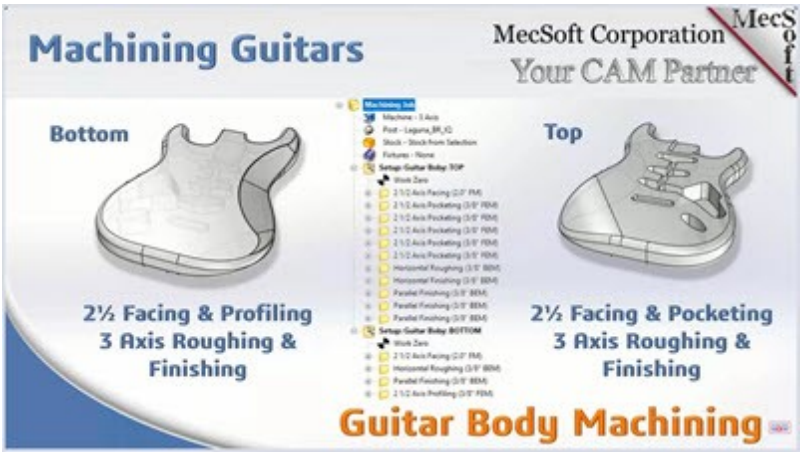
- Match cutter size carefully to mold detail requirements.
- Reduce stepover values to improve mold surface finish quality.
- Validate machining workflows with simulation before machining.

4.32 Machining Guitars

Link: https://www.youtube.com/watch?v=zwKNwR_A3BQ

This open AMS webinar focuses on practical tips and machining techniques for manufacturing guitars using RhinoCAM and VisualCAD/CAM. The presentation walks through real-world workflows used by guitar builders and CNC programmers for machining guitar bodies, necks, cavities, contours, and decorative features. Topics include Fixturing, two-sided machining, roughing, finishing, tooling selection, woodworking considerations, simulation, and post processing. Throughout the webinar, the presenters also share practical Luther-focused machining tips for improving surface finish quality, reducing tear-out, simplifying setups, and streamlining CNC guitar manufacturing workflows.

Source Files: 20.21.08 Open Webinar Tips & Techniques for Machining Guitars.zip



1 of 19 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of guitar components and webinar goals
5:00	Guitar Body Machining – Setup & Top Side	Square stock with diagonal locating pins, facing, pocketing (neck pocket & controls), parallel finishing

Time	Topic / Section	Description
18:00	Guitar Body – Bottom Side & Flip Machining	Flipping the part, facing, roughing contoured areas, horizontal finishing with Z containment, final profiling to separate from stock
28:00	Locked Operations	How to lock and archive alternative toolpaths (e.g., opposite-direction finishing)
32:00	Guitar Neck Machining Challenges & Solutions	Why standard 3-axis finishing doesn't reach bottom edges; introduction of catch planes/surfaces
38:00	Neck Boundary Curve Creation	Using Rhino commands (Duplicate Edge, Project, Curve Boolean) to create clean outer boundaries + offset by tool radius
43:00	Edge Protection with Extruded Surfaces	Preventing tool drop-off and damage to sharp neck edges
48:00	Pre-Cut Blank Machining (Dulcimer Example)	Overlapping offset boundaries, using parallel finishing as roughing, catch surfaces for laminated areas
53:00	Inlay Machining (Mother of Pearl)	Pocketing the female inlay, 2.5-axis re-machining for corner cleanup, new 2021 corner relief (dog bones)
58:00	Fret Slotting with Engraving Operation	Using engraving with multiple Z step-downs on curved fretboard surfaces
1:00:00	Q&A and Wrap-Up	Final tips and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Catch Planes / Catch Surfaces Are Game-Changers for Necks** — When standard 3-axis finishing won't reach the bottom edges of a neck (because the tool center-line aligns with the surface edge), create a temporary catch plane or surface below the part. This allows the tool to drop fully and machine clean edges.
2. **Offset Boundary Curves by Tool Radius** — Always offset your outer control curve outward by the tool radius + clearance. This ensures the tool reaches the true edge without plunging into the spoil-board or leaving uncut material.

3. **Use Rhino Commands for Clean Boundaries** — Combine Duplicate Edge → Project → Curve Boolean to create a single, continuous, closed outer boundary curve from complex 3D geometry. This is far more reliable than manual curve creation.
4. **Extruded Edge Protection Surfaces** — Add thin extruded surfaces along the back and sides of necks (or any sharp edges) to prevent the tool from dropping off and damaging delicate features during finishing.
5. **Parallel Finishing as Roughing** — On pre-cut or laminated blanks, use parallel finishing with multiple step-downs as a roughing operation. It leaves uniform stock for a final finish pass and works surprisingly well on contoured areas.
6. **Locked Operations for Alternative Toolpaths** — Lock operations you don't want to post (e.g., a second finishing pass in the opposite direction). This keeps them in the tree for future reference or quick swapping without deleting them.
7. **2.5-Axis Re-Machining for Inlays** — After rough pocketing with a larger tool, use the Re-Machining feature with a smaller tool to automatically clean up corners. This is much faster than manually creating additional operations.
8. **Corner Relief (Dog Bones)** — In profiling operations, enable internal dog-bone corner relief to automatically create clearance for square inlays without manually modeling the relief.
9. **Engraving for Curved Fret Slots** — Use the Engraving operation with the tool tip following a curve drawn on the fretboard surface. Add multiple Z step-downs to reach full depth. This allows perfectly contoured fret slots that follow the fretboard radius — no more gaps from straight cuts.
10. **Diagonal Locating Pins for Flip Machining** — For guitar bodies, use two diagonal locating pins in the stock. This provides excellent registration when flipping the part and is more reliable than edge stops for contoured shapes.
11. **Z Containment for Flip Machining Safety** — On the bottom side, use Z containment to prevent the tool from accidentally machining into the already-finished top face.

Key Takeaways

This webinar is one of the best real-world guitar machining resources available. It shows how to solve common challenges (neck edge finishing, flip registration, inlay corners, curved fret slots) using clever geometry tricks and underused 2.5-axis features.

Biggest lessons:

- Catch planes/surfaces + properly offset boundaries solve most "tool won't reach the edge" problems.
- Re-machining and corner relief features (especially 2021+) dramatically speed up inlay work.

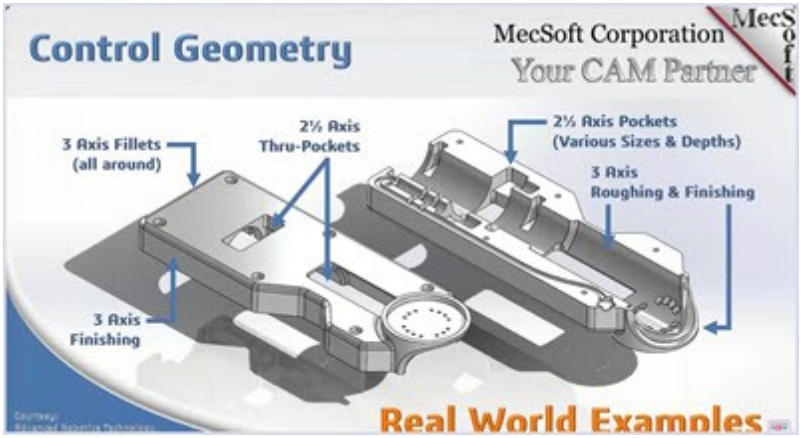
- Engraving with step-downs is perfect for curved fret slots.
- Plan guitar Fixturing and setups before creating toolpaths.
- Use dowel pins and registration systems for accurate flip machining.
- Preserve holding material until final machining operations.
- Use sharp woodworking cutters to reduce tear-out and sanding.
- Combine 2½ Axis and 3 Axis workflows for efficient guitar machining.
- Verify setups and machining operations using simulation before cutting material.

4.33 Control Geometry

Link: https://www.youtube.com/watch?v=MXz5DSi-tEI

This AMS webinar focuses on the effective use of control geometry in RhinoCAM and VisualCAD/CAM. The presentation explains how control geometry can be used to guide, limit, contain, and optimize machining operations for both 2½ Axis and 3 Axis workflows. Topics include machining boundaries, containment curves, drive geometry, avoidance regions, cut direction control, region selection, roughing optimization, finishing control, simulation, and workflow troubleshooting. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining accuracy, reducing unnecessary machine motion, and creating safer and more efficient CNC machining workflows.

Source Files: 20.21.07 AMS Webinar The Effective Use of Control Geometry.zip



1 of 12 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Why Control Geometry Matters	Common mistakes and the importance of proper curve setup

Time	Topic / Section	Description
4:30	What is Control Geometry?	Curves, regions, containment, avoid regions, and catch planes
9:15	Closed vs Open Curves & Curve Direction	How direction and start point affect cutting side (left/right rule)
14:40	Predefined Regions & Cloning	Creating named regions for fast selection and reuse
20:10	Merging Curves & Fixing Gaps	Best practices for clean, continuous profiles
25:50	Offset Curves & Containment	When and how to offset for safe finishing near edges
32:00	Avoid Regions & Catch Planes/Surfaces	Protecting areas and extending tool reach on contoured parts
40:30	**Live Demo** – Guitar Neck & Complex Pocket	Real-world application of all concepts
52:00	Common Mistakes & Troubleshooting	Over-cutting, tool wrapping, and how to fix them
58:00	Q&A and Best Practices Summary	Final recommendations and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Curve Direction Determines Cutting Side** — Always check the direction and start point of your control curves. The software cuts to the **left** of the curve direction by default for closed profiles.
2. **Merge Curves Before Machining** — Never use multiple separate curves for a single profile. Always merge them into one continuous entity. This prevents gaps, incorrect start points, and broken toolpaths.
3. **Offset Curves Outward for Safe Finishing** — When finishing near edges or parting lines, offset your containment curve **outward** by the tool radius + clearance. This prevents the tool from rolling off the edge or gouging.

4. **Predefined Regions Save Massive Time** — Create and name regions in the Machining Browser for hole groups, pockets, or profiles. You can then quickly select them in future operations or clone entire jobs.
5. **Avoid Regions Must Bisect the Geometry** — An Avoid Region only works if it **crosses** (bisects) the control geometry. Simply overlapping is not enough.
6. **Catch Planes/Surfaces for Edge Reach** — When the tool won't reach the bottom edge of a contoured part (common on guitar necks or mold cores), create a temporary catch plane or surface below the part so the tool can fully drop and machine the edge cleanly.
7. **Use "Pick Top" for Flexible Depth Control** — When your control geometry is at the bottom but you want to start machining from the top of stock, use the **Pick Top** option instead of guessing Z values.
8. **Closed Curves for Pocketing & Facing** — Pocketing and facing require **closed** curves. The software can virtually close open pockets, but it's more reliable to create proper closed boundaries.
9. **Clone Operations Instead of Recreating** — After perfecting one operation, clone it and simply reassign new control geometry. This is much faster than starting from scratch.
10. **Hide Unnecessary Geometry** — Turn off layers or hide geometry not used in the current operation before generating toolpaths. This speeds up calculation significantly.
11. **Test Curve Direction Early** — Always simulate a single profiling or pocketing operation first to verify the tool is cutting on the correct side before committing to the full job.

Key Takeaways

This webinar is one of the **best resources** for mastering control geometry — the foundation of all 2.5-axis work in MecSoft CAM.

Biggest lessons:

- The techniques shown remain fully relevant and are still used daily in current versions of RhinoCAM and VisualCAD/CAM.
- Proper curve preparation (merging, direction, offsetting) prevents 80% of toolpath problems.
- Predefined regions + cloning = huge time savings on repetitive jobs.
- Catch planes/surfaces are essential for reaching difficult edges on contoured parts.
- Organize control geometry on dedicated layers for easier management.
- Use containment regions to reduce unnecessary air cutting.
- Simplify control curves to improve machining stability and regeneration speed.
- Verify curve direction and machining boundaries before creating toolpaths.

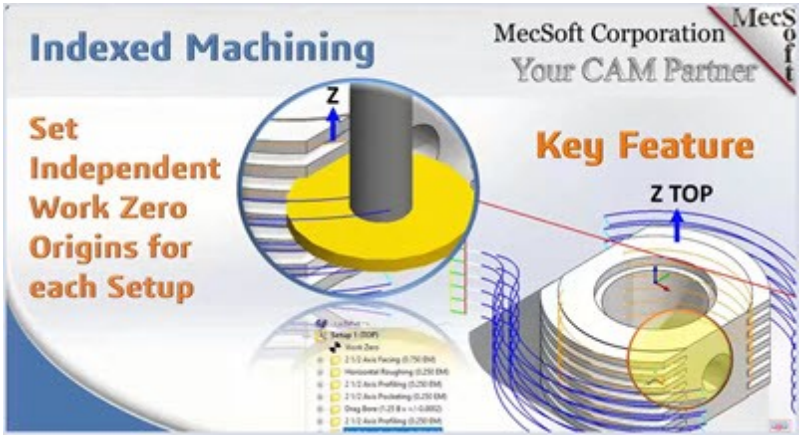
- Use avoidance geometry to protect fixtures and non-machined areas.
- Apply control geometry strategically in both 2½ Axis and 3 Axis workflows.

4.34 5 Axis Indexed

Link: https://www.youtube.com/watch?v=i91_IulXGUQ

This AMS webinar focuses on multi-axis indexed machining workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how indexed machining uses rotary positioning to machine multiple sides of a part while still relying on standard 3-axis machining strategies. Topics include rotary setup definition, indexed positioning, machining coordinate systems, setup orientation, roughing and finishing operations, simulation, post processing, and workflow optimization. Throughout the webinar, the presenters also share practical CAM programming tips for simplifying complex part machining, reducing setup time, and improving machining accuracy using indexed multi-axis workflows.

Source Files: 20.21.05 AMS Webinar Multi-Axis Indexed Machining.zip



1 of 16 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Benefits	Why indexed machining improves accuracy and reduces errors
5:00	Coordinate Systems Explained	Machine, World, and Work Coordinate Systems + Work Zero Offsets
10:00	3-Axis Indexed Machining	XY Instancing, Polar Instancing, Fixture Offset Programming (G54)

Time	Topic / Section	Description
25:30	4-Axis Indexed Machining (3+1)	Flip Machining + 4th Axis Rotary Instancing
35:20	5-Axis Indexed Machining (3+2)	General multi-sided machining + Tombstone machining
49:00	Q&A, Catch Planes & Best Practices	Tool change points, collision avoidance, and final tips

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Indexed Machining = One File, Multiple Setups** — All sides of the part are programmed in a single part file. This eliminates manual alignment errors and allows full simulation and posting of the entire job at once.
2. **Use Setups to Define Work Coordinate Systems** — Create a new Setup for each orientation. The software automatically handles the coordinate transformation when you rotate the part or machine axes.
3. **Catch Planes Prevent Over-Cutting** — When flipping a part (especially on contoured surfaces), create a temporary CAD surface (Catch Plane) below the part to stop the tool from machining too deep on the second side.
4. **Reselect Control Geometry After Copying Operations** — When copying operations to a new setup, the original control geometry often stays linked. Always reselect the correct geometry for the new orientation.
5. **Tool Change Point for Safe Rotation** — Define a safe Tool Change Point in the machine setup so the tool automatically retracts before the table rotates. This prevents collisions during indexed moves.
6. **Fixture Offsets (G54/G55) for Multiple Parts** — Instead of copying toolpaths, create multiple Work Zeros with different X/Y/Z offsets. The post automatically outputs the correct G54/G55 commands.
7. **Rotary Instancing for Radial Features** — For parts with identical features around a cylinder, program one feature and use Rotary Instance to automatically generate all copies with the correct 4th-axis rotation.
8. **Face Top Operation (2021 Feature)** — Use the new “Face Top” command to automatically detect and machine the top face of a setup without manually selecting control geometry.

9. **Professional Configuration Required for 5-Axis Indexed** — Multi-sided (3+2) and Tombstone machining require the Professional configuration.
10. **Post-Processor Must Support Indexed Output** — Make sure your post is configured to output the correct work offsets and rotary commands for your machine.

Key Takeaways

This webinar clearly explains why Indexed Machining is often more practical and reliable than continuous 4/5-axis for many real-world parts:

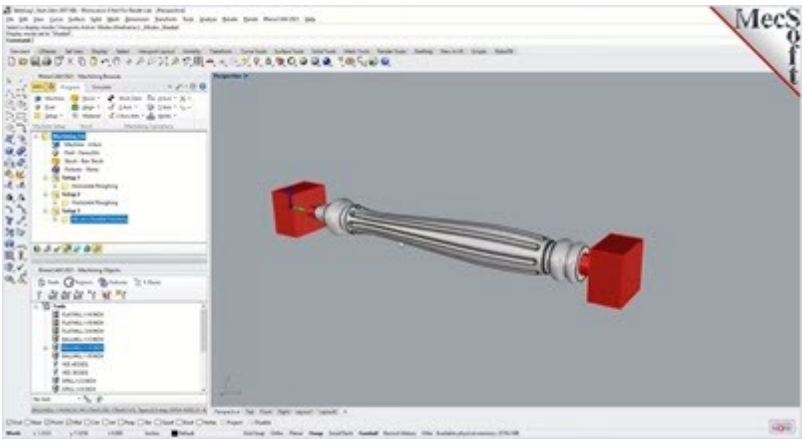
- Best for: Parts with features on multiple flat or perpendicular sides (cylinder heads, brackets, tombstone fixtures, flip-machined parts, etc.).
- Higher accuracy (uses machine's positioning precision)
- Easier to program and simulate
- All operations stay in one file
- Works great on 4-axis and 5-axis machines with rotary tables or heads
- Use indexed machining to simplify multi-sided part programming.
- Verify rotary setup alignment before creating toolpaths.
- Reuse proven 3-axis machining operations across indexed orientations.
- Simulate every indexed setup before machining production parts.
- Organize machining coordinate systems carefully for multi-axis workflows.
- Reduce setup complexity by planning indexed orientations early.

4.35 4 Axis Introduction

Link: https://youtu.be/iHS-4IQThKw

This AMS Webinar introduces 4-Axis Machining in MecSoft's RhinoCAM and VisualCAD/CAM. The session explains part orientation, machine setup, indexed vs continuous strategies, common machining methods, and post-processing. It uses practical examples including multi-side indexing and cylindrical wrap/projection operations to demonstrate real workflows.

Source Files: 20.21.02 AMS Webinar Introduction to 4 Axis Machining.zip



1 of 16 Slides Total

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar overview
3:30	4-Axis Machining Basics	Definition of 4-axis milling, rotary axis (A/B) configuration, and machine setup requirements.
7:00	Part Orientation and Setup	Setting rotary axis alignment, origin placement, and rotation center for accurate indexing.
12:00	Machining Workflow Overview	Loading part/stock, defining machine configuration, selecting post, and planning operations.
17:00	Indexed 4-Axis Machining Demo – Part 1	Top face operations (pocketing, drilling, chamfering) at 0° using Rotate Table Setup.
25:00	Indexed 4-Axis Machining Demo – Part 2	Side machining at 90°, 180°, and 270° rotations with drilling and profiling.
33:00	3-Axis Operations in Indexed Setup	Horizontal roughing from multiple sides on a table leg example with stock-to-leave control.
40:00	Continuous 4-Axis – Wrap Methods	4-Axis facing, pocketing, and profiling on cylindrical parts using wrap toolpaths.

Time	Topic / Section	Description
48:00	Continuous 4-Axis – Projection Methods	Parallel finishing with “Along”, “Across”, and Helical cut patterns on indexed parts.
55:00	4-Axis Engraving	Projecting text and curves onto cylindrical surfaces with step-down engraving toolpaths.
60:00	Cut Pattern & Containment Options	Adjusting cut patterns, step-over, and axial containment for better results.
63:00	Q&A and Best Practices	Indexed vs continuous recommendations, simulation tips, and final questions.

Key Tips & Best Practices Extracted

1. **Set rotation center at workpiece center** — Place the rotation center at the center of the part for accurate and predictable indexing across multiple setups.
2. **Use Rotate Table Setup for indexing** — Create separate setups at 0°, 90°, 180°, and 270° to machine all sides without re clamping the part.
3. **Prefer indexed machining for roughing** — Indexed methods provide better rigidity and tool life during heavy material removal.
4. **Use continuous methods for finishing** — Projection and wrap strategies deliver superior surface finish on curved or cylindrical features.
5. **Define rotary axis correctly in machine setup** — Specify whether the rotary axis rotates around X or Y and match it to your actual CNC configuration.
6. **Apply stock-to-leave on indexed roughing** — Leave a small amount of stock when roughing from opposite sides to avoid visible seams after finishing.
7. **Choose the right cut pattern in continuous mode** — Use “Along” for speed or “Across”/Helical for better surface finish on cylindrical parts.
8. **Set axial containment limits** — Restrict toolpaths along the rotary axis to prevent unnecessary air cuts and reduce cycle time.
9. **Simulate all setups before posting** — Always run full simulation across every indexed angle to catch collisions or missed areas.
10. **Select a post processor that supports rotary output** — Ensure your post outputs correct A-axis (or B-axis) angles and continuous rotation codes.

Key Takeaways

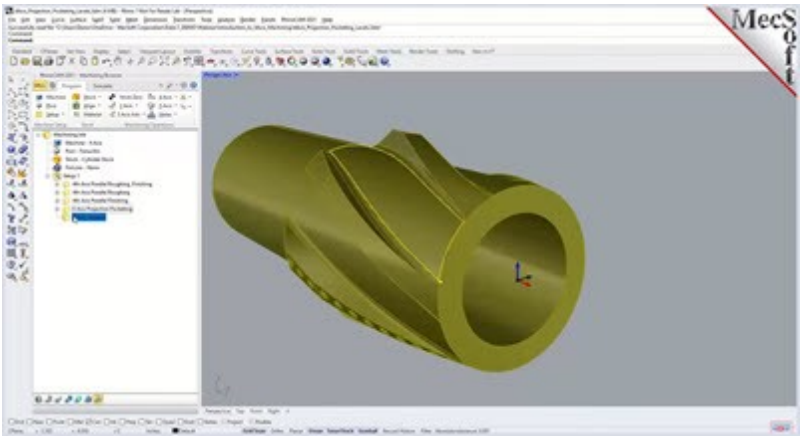
- 4-Axis machining allows multi-side and cylindrical work without multiple setups by using a rotary axis.
- Proper part orientation and rotation center placement are critical for accurate indexed machining.
- Indexed strategies are best for roughing due to rigidity, while continuous methods excel at finishing.
- Wrap methods are ideal for constant-radius cylindrical features, while projection methods work well for complex surfaces.
- Simulation across all rotary positions helps catch issues before posting G-code.
- Cut pattern choice (“Along”, “Across”, or Helical) and axial containment significantly affect surface finish and cycle time.
- MecSoft’s Rotate Table Setup makes programming indexed 4-axis parts straightforward and organized.

4.36 4 Axis Advanced

Link: https://www.youtube.com/watch?v=vzBOiaqX7y0

This AMS webinar focuses on advanced 4-axis machining workflows in RhinoCAM and VisualCAD/CAM. The presentation explores rotary machining techniques used for complex cylindrical and wrapped geometry, including indexed and continuous rotary machining strategies. Topics include machine setup, rotary axis configuration, stock definition, roughing and finishing operations, tool orientation, collision avoidance, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving rotary machining accuracy, reducing cycle times, and creating efficient advanced 4-axis machining workflows for real-world CNC applications.

Source Files: 20.21.03 AMS Webinar Replay 4 Axis Advanced Machining.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of advanced 4-axis capabilities
3:45	Machine Setup & 4-Axis Definition	Rotary axis orientation, stock setup, origin placement
8:20	4-Axis Facing (Cylindrical Surfaces)	Facing between offset curves on wrapped geometry
14:10	4-Axis Engraving	Tool tip following curves on cylindrical features
18:45	4-Axis Profiling	Edge cleanup and material removal on cylinder ends
23:30	Drive Surface Machining (True 4-Axis)	Simultaneous 4-axis motion — tool stays normal to surface
35:00	Projection Pocketing	Machining grooves not perpendicular to the rotary axis
42:15	Safety Options & Tool Change Points	Auto-lift, collision avoidance, post-processor setup
50:00	Live Demo & Simulation Review	Full job simulation with coordinated X/Y/Z/A moves
56:00	Q&A and Best Practices	Final recommendations and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Drive Surface Machining = True Simultaneous 4-Axis** — This is one of the few operations where the tool, linear axes, **and** rotary axis all move together. It requires manually creating a drive surface (not a simple wrap) and is essential for non-centered or overhanging geometry.
2. **Offset Curves by Tool Radius** — When facing or engraving on cylindrical features, always offset your control curves outward by the tool radius to prevent gouging into the main stock.

3. **Use Masking Surfaces** — Create masking surfaces to maintain stock connections during wrapping operations and prevent the tool from cutting into areas you want to protect.
4. **Projection Pocketing for Non-Normal Features** — When grooves or features are not perpendicular to the rotary axis (e.g., helical or skewed), use **Projection Pocketing** to project the toolpath onto the underlying surface while keeping the tool normal to the rotary axis.
5. **Auto-Lift During Rotation** — Enable the safety option to automatically lift the tool to the tool change position before the table rotates. This prevents collisions.
6. **Define Tool Change Point** — Set a safe tool change point in General Parameters so the post automatically inserts a retract move between setups.
7. **Engraving Only Works When Curves Are Normal** — 4-axis engraving assumes the curves are normal (perpendicular) to the rotary axis. For skewed features, use Projection Pocketing instead.
8. **Flute Length Matters** — Make sure your tool has enough flute length to reach the widest material section when the part is rotated.
9. **Verify Post-Processor Support** — Not all posts handle advanced 4-axis motion correctly. Test thoroughly or request a custom post from MecSoft support.
10. **Simulate Every Operation** — Especially important in 4-axis work — check tool orientation and collision detection after every setup change.
11. **Part Files Available** — The webinar provides down-loadable part files (link in the video description) so you can follow along and practice.

Key Takeaways

This webinar is excellent for users who want to move beyond basic 3+1 indexing into **true continuous 4-axis machining**. The standout techniques are:

- Best for: Users with 4-axis machines who want to machine more complex geometries efficiently while maintaining good surface finish and tool life.
- Drive Surface Machining — For complex simultaneous 4-axis motion
- Projection Pocketing — For features not aligned with the rotary axis
- Smart use of offset curves and masking — To protect stock and achieve clean finishes
- Use indexed machining whenever simultaneous rotary motion is unnecessary.
- Verify rotary alignment carefully before generating toolpaths.
- Optimize stepover settings for improved rotary surface finish quality.
- Simulate rotary movement before posting NC code.
- Monitor machine limits and collision zones during advanced rotary machining.

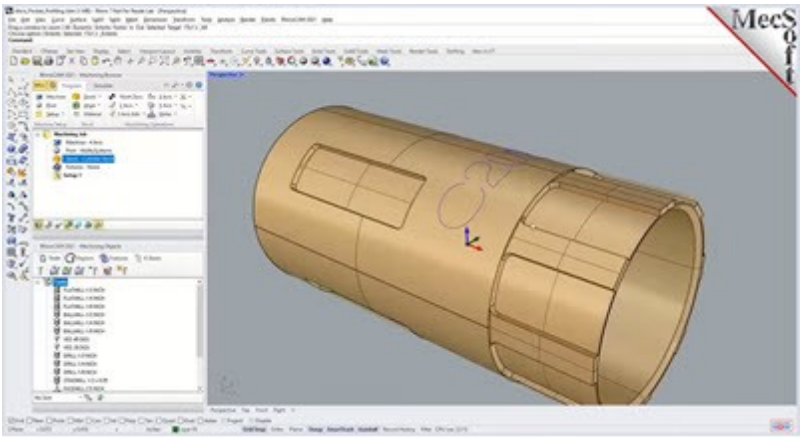
- Organize setups carefully to simplify multi-orientation workflows.

4.37 4 Axis Basic

Link: https://www.youtube.com/watch?v=iHS-4IQThKw

This AMS webinar introduces the fundamentals of 4-axis machining in RhinoCAM and VisualCAD/CAM. The presentation explains how rotary machining expands CNC machining capabilities by allowing the part to rotate around an additional axis during machining. Topics include rotary setup configuration, indexed and continuous 4-axis workflows, wrapped toolpaths, stock definition, roughing and finishing operations, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving setup accuracy, simplifying rotary machining workflows, and achieving efficient 4-axis machining results on cylindrical and wrapped geometry.

Source Files: x.x.x AMS Webinar: Configurations



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview of 4-axis capabilities
4:07	4-Axis Setup and Configuration	Machine tool setup, table vs head, rotational axis (X or Y), work origin, rotation center
13:00	Indexed 4-Axis Machining Demo 1 (Prismatic Part)	Top face operations + Rotate Table Setup at 90°, 180°, 270° with copy/paste operations
26:00	Indexed 4-Axis Machining Demo 2 (Y-Axis Flip)	180° flip machining with Horizontal Roughing on top and bottom faces

Time	Topic / Section	Description
31:50	Continuous 4-Axis: Wrap Methods	4-Axis Facing, Pocketing, and Profiling on cylindrical stock
40:00	Continuous 4-Axis: Projection Methods	4-Axis Parallel Finishing ("Along Axis" vs "Across Axis"), Cut Axial Containment, Start/End Angle, 4-Axis Engraving
51:50	Q&A and Best Practices	Cut pattern choice, containment settings, and closing remarks

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Indexed Machining = One File, Multiple Setups** — Use "Rotate Table Setup" to create new setups at fixed angles (0°, 90°, 180°, 270°). Copy and paste operations from the first setup — the software automatically transforms the toolpaths for the new orientation.
2. **Work Origin & Rotation Center Must Match** — Always set both the work origin and the rotation center at the same point (usually 0,0,0 or the center of the stock) for accurate coordinate output.
3. **Limit Containment Slightly Below Zero on Flip Machining** — When machining the bottom face after a 180° flip, set bottom containment slightly below Z=0 to prevent a visible seam between top and bottom roughing passes.
4. **Wrap Methods for Simple Cylindrical Features** — Use 4-Axis Facing, 4-Axis Pocketing, and 4-Axis Profiling when features are on a constant-radius cylinder. These "unwrap" the geometry for easy 2.5-axis-style programming.
5. **Projection Methods for Complex Surfaces** — Use 4-Axis Parallel Finishing when the part has varying radius or complex surfaces. Choose "Along Axis" (faster, table rotates incrementally) or "Across Axis" (slower but continuous rotation, often better finish).
6. **Cut Axial Containment + Start/End Angle** — These two powerful settings let you restrict machining to specific sections of a cylinder (e.g., only machine between -90° and +90° or between specific Z heights).
7. **Indexed for Roughing, Continuous for Finishing** — The recommended workflow: use indexed methods for roughing (more rigid, faster material removal) and continuous methods for final finishing (better surface quality).

8. **4-Axis Engraving Projects Curves onto the Cylinder** — Simply select the text/curves and the cylindrical surface — the software automatically wraps the toolpath at the correct depth.
9. **Post-Processor Must Output A-Axis Angles** — Make sure your post is configured to output the rotary axis (usually A) with the correct rotation direction and format.
10. **Expert Configuration Required** — Most 4-axis features (especially continuous wrapping and projection) require the Expert configuration or higher.

Key Takeaways

This webinar is the perfect starting point for anyone new to 4-axis machining in MecSoft CAM. It clearly shows:

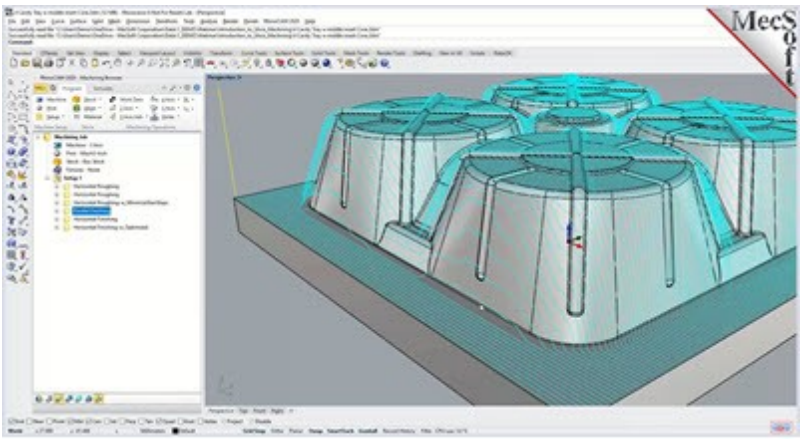
- The live demos on both a prismatic part and a cylindrical part make the concepts very easy to understand and apply immediately.
- How to set up a 4-axis machine (table configuration, origin, rotation center).
- The difference between Indexed (simple, rigid) and Continuous (more flexible, better finish).
- Practical workflows using Rotate Table Setup, Wrap methods, and Projection methods.
- Start with indexed machining before learning continuous 4-axis workflows.
- Verify rotary alignment carefully before generating toolpaths.
- Apply standard 3-axis machining knowledge within rotary machining environments.
- Simulate rotary movement before posting NC code.
- Use wrapped machining workflows for cylindrical geometry applications.
- Organize setups carefully to simplify rotary machining operations.

4.38 3 Axis Power Topics!

Link: https://www.youtube.com/watch?v=f2hVIawXCSI

This AMS webinar focuses on advanced “power topics” in 3-axis machining using RhinoCAM and VisualCAD/CAM. The presentation explores high-level machining workflows and optimization strategies for machining complex 3D geometry efficiently and accurately. Topics include advanced roughing, re-roughing, finishing strategies, tool containment, stepover optimization, cutter selection, machining regions, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving surface finish quality, reducing cycle time, minimizing air cutting, and creating more efficient 3-axis machining workflows for real-world CNC applications.

Source Files: 20.20.11 AMS Webinar 3 Axis Power Topics.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of 9 power topics for 3-axis machining
2:15	Role of Tolerances	How tolerance affects cord segments, arc fitting, and G-code size
11:23	Effect of Steep/Flat Areas on Toolpath Behavior	Parallel vs Horizontal finishing — when to use each
17:23	Containing the Cutter	Offset containment, guide surfaces, skim clearance, and preventing erratic retracts
33:16	Advanced Stock Creation Methods	Part Offset Stock, Stock from Selection, Stock from Extrusion (Region Stock)
41:02	Minimize Stair Steps in Horizontal Roughing	New feature that dramatically reduces stair-step artifacts efficiently
48:30	Optimized Machining in Horizontal Finishing	“Optimized XY Machining” — combining Z-level and parallel strategies
50:50	Pre-drill Start Points	Using pre-drilled holes to define exact entry locations for roughing
51:12	Setting Default Parameters using Knowledge Bases	Saving and loading custom parameter sets for different materials/tools

Time	Topic / Section	Description
59:03	Toolpath Editor Tools	Global edits (transform) and Selection edits (delete sections) — Pro/Premium only

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Tolerance = Speed vs Quality Trade-off** — Tighter tolerances (e.g., 0.001") produce smoother finishes but create much larger G-code and longer calculation times. Use 0.005"–0.01" for roughing and only tighten for final finishing. Enable **Arc Fitting** (set to ~2× global tolerance) to convert linear moves to G2/G3 arcs for smoother motion and smaller files.
2. **Parallel Finishing for Flat Areas, Horizontal for Steep Areas** — Using parallel finishing on steep walls causes stair-stepping and jerky motion. Use **Horizontal (Z-level) Finishing** for steep geometry and **Parallel Finishing** for flat or gently contoured areas. For mixed parts, run multiple operations or use the new **Optimized Horizontal Finishing**.
3. **Offset Containment by Tool Radius** — To ensure the tool reaches the full depth on vertical walls or fillets, offset your containment curve **outward** by at least the tool radius + clearance. This prevents the tool from stopping short or retracting prematurely.
4. **Use Guide Surfaces for Difficult Containment** — When no underlying geometry exists (e.g., open pockets or edges), insert a temporary planar or sculpted surface to guide the tool and eliminate erratic retracts. Set "Skim" clearance plane height to reduce unnecessary Z moves.
5. **Advanced Stock Methods Save Huge Time**:
    - **Part Offset Stock** — Perfect for casting workflows (creates stock as an offset of the finished part).
    - **Stock from Selection** — Use watertight poly surfaces/meshes defined in CAD as exact stock.
    - **Stock from Extrusion (Region Stock)** — Extrude closed planar curves to create stock with holes — ideal for nested or plate stock.
6. **Minimize Stair Steps Feature is a Game-Changer** — Instead of using very small step-downs for the entire roughing operation (which explodes calculation time), use this feature: it roughs with larger steps first, then automatically cleans up the remaining stair steps in a secondary pass. Much faster and cleaner results.
7. **Optimized XY Machining in Horizontal Finishing** — Enable this option to let Z-level finishing also generate offset parallel passes on flat areas. This gives you the best of both worlds in a single operation.

8. **Pre-drill Start Points for Safe Entry** — Select a pre-drilled hole in the **Start Points** tab of roughing operations. The tool will ramp or plunge exactly at that location — excellent for deep cavities or when you want to control entry precisely.
9. **Knowledge Bases = Instant Defaults** — Save complete parameter sets (tolerances, feeds, step-over, clearance, etc.) as named Knowledge Bases. You can then load the correct one automatically based on material, tool size, or part type. Huge time saver for consistent programming.
10. **Toolpath Editor (Pro/Premium)** — After generating a toolpath, use **Global Edits** to move, rotate, mirror, or scale the entire path (great for refinish passes). Use **Selection Edits** to delete specific sections (perfect for resuming a job after tool breakage without regenerating everything).

Key Takeaways

This webinar is packed with **high-impact techniques** that every intermediate-to-advanced 3-axis user should know. The biggest productivity boosters are:

- Many of these features (especially Minimize Stair Steps and Optimized XY Machining) were relatively new at the time but remain extremely useful in current versions of RhinoCAM and VisualCAD/CAM.
- Proper containment + guide surfaces
- Minimize Stair Steps + Optimized Horizontal Finishing
- Knowledge Bases for consistent defaults
- Toolpath Editor for post-generation fixes
- Organize machining regions before creating advanced toolpaths.
- Use re-roughing and rest machining to improve finishing efficiency.
- Balance surface finish quality against machining cycle time.
- Use containment boundaries to reduce unnecessary air cutting.
- Match cutter geometry carefully to surface detail requirements.
- Verify machining workflows with simulation before posting code.

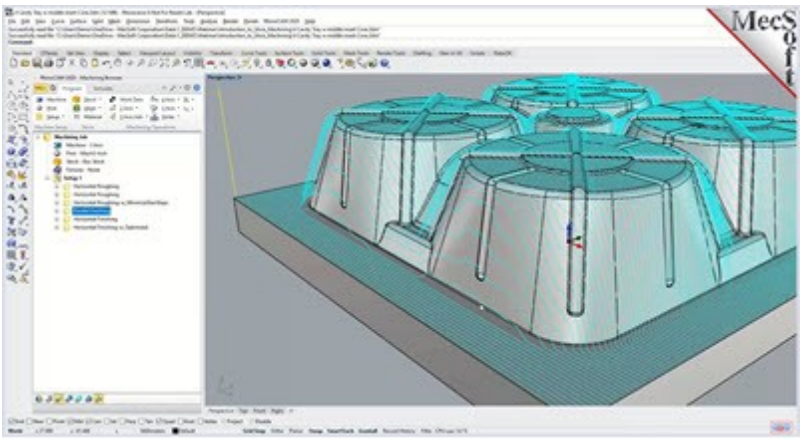
4.39 3 Axis Advanced

Link: https://www.youtube.com/watch?v=uEWRH7Fon34

This AMS webinar focuses on advanced 3-axis machining workflows in RhinoCAM and VisualCAD/CAM. The presentation explores advanced roughing and finishing techniques used to machine complex 3D parts efficiently and accurately. Topics include setup preparation, machining regions, horizontal roughing, re-roughing, finishing strategies, rest machining, cutter selection, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving surface finish quality, reducing cycle times, minimizing air

cutting, and optimizing advanced 3-axis machining workflows for real-world manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of advanced 3-axis strategies for molds and complex parts
4:30	Horizontal Re-roughing	Removing in-process stock left by larger roughing tools using smaller tools in tight corners
13:00	Flat Area Machining (Clear Flats & Plateau)	Automatic detection and machining of flat or near-flat areas (up to a user-defined angle)
22:00	Radial Machining	Zigzag or single-direction finishing for circular/near-circular features (inside-out or outside-in)
27:30	Spiral Machining	Continuous spiral toolpaths for circular regions with user-defined center point
32:00	3D Offset Pocketing (Constant 3D Step Over)	Maintains uniform scallop height across steep and shallow areas for consistent finish
40:00	Pencil Tracing	Automatic cleanup of bi-tangent areas (sharp corners and fillets) using reference tool diameter

Time	Topic / Section	Description
46:00	Valley Machining	Re-machining areas left by larger previous tools using a smaller reference diameter
51:00	Curve-Based Machining (Projection & Between Curves)	Driving toolpaths along projected curves or flowing between two curves
56:00	Surface Extent Conditions (On / To / Past)	Controlling how toolpaths behave at surface boundaries
58:00	Q&A and Best Practices Wrap-Up	Final tips and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Horizontal Re-roughing Saves Massive Time** — After roughing with a large tool, use Horizontal Re-roughing with a smaller tool and zero stock allowance. It automatically removes the remaining "stair-step" stock in corners without manually creating new containment geometry.
2. **Clear Flats & Plateau Machining Eliminate Manual Selection** — Enable "Clear Flats" or use Plateau (Flats) machining with a maximum slope angle (e.g., 35°). The software automatically identifies and machines all flat or near-flat areas — no need to manually select regions.
3. **3D Offset Pocketing (Constant Scallop) for Superior Finish** — This operation maintains a constant 3D step-over across both steep and shallow areas, delivering a much more uniform surface finish than traditional parallel or horizontal finishing on complex molds.
4. **Pencil Trace for Automatic Corner Cleanup** — Use Pencil Trace with a ball mill or corner radius tool. It automatically finds bi-tangent lines (where the tool touches two surfaces at once) and cleans up sharp internal corners and fillets that standard finishing misses.
5. **Valley Machining for Efficient Re-machining** — After roughing with a large tool, use Valley Machining with a smaller reference diameter. It automatically identifies and re-cuts the uncut "valleys" left behind — extremely effective for mold cavities.
6. **Radial & Spiral Machining for Circular Features** — Use Radial for quick zigzag coverage of round areas and Spiral when you want a single continuous toolpath with no retracts. Both are available in Standard configuration.

7. **Curve Projection Machining for Complex Boundaries** — Project 2D curves onto 3D surfaces to drive toolpaths exactly where you need them (great for engraving or following irregular boundaries).
8. **Between Curve Machining Requires Careful Alignment** — When using "Between Two Curves," make sure both curves have consistent direction and matching start points. Misalignment causes erratic toolpaths.
9. **Surface Extent Conditions Give Precise Control** — Use "On" for exact boundary following, "To" to stop at the surface edge, or "Past" to allow the tool to continue slightly beyond the surface for clean tangent finishes.
10. **Combine Techniques for Best Results** — Typical high-quality mold workflow: Rough → Re-rough → Clear Flats/Plateau → 3D Offset or Parallel Finishing → Pencil Trace/Valley for final cleanup.

Key Takeaways

This webinar is packed with high-impact advanced techniques that dramatically improve both programming speed and surface quality on complex 3-axis parts (especially molds).

Biggest productivity boosters highlighted:

- These techniques remain highly relevant and are still widely used in current versions of RhinoCAM and VisualCAD/CAM.
- Horizontal Re-roughing + Clear Flats/Plateau = massive time savings on stock removal.
- 3D Offset Pocketing (Constant Scallop) = superior, consistent finish.
- Pencil Trace + Valley Machining = automatic, intelligent corner and valley cleanup.
- Curve Projection & Between Curves = precise control on irregular geometry
- Organize complex geometry into manageable machining regions.
- Use re-roughing and rest machining to improve finishing efficiency.
- Apply curve-based machining strategies for better toolpath control.
- Use pencil tracing and valley re-machining for fine detail cleanup.
- Optimize roughing workflows before refining finishing operations.
- Validate machining operations using simulation before posting code.

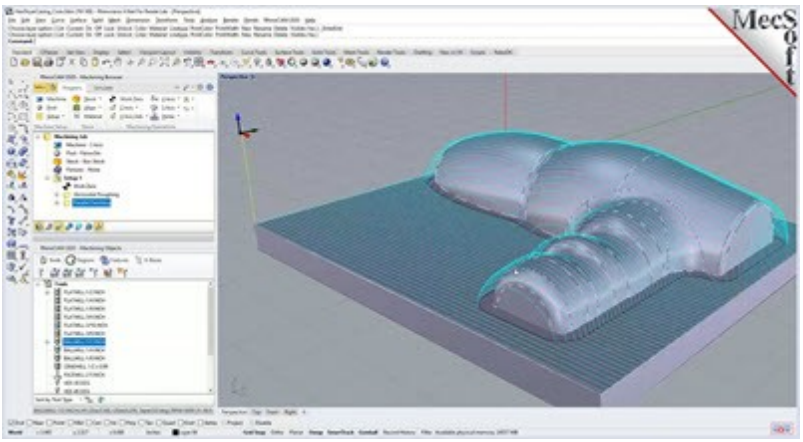
4.40 3 Axis Bsaic

Link: https://www.youtube.com/watch?v=7iHZH52YYHo

This AMS webinar introduces the fundamentals of 3-axis machining in RhinoCAM and VisualCAD/CAM. The presentation walks through the core concepts and workflows used to machine 3D parts using standard XYZ machining motion. Topics include geometry preparation, stock setup, roughing operations, finishing strategies, cutter selection, stepover control,

simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining efficiency, achieving better surface finishes, reducing setup errors, and building reliable 3-axis machining workflows for real-world CNC applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of 3-axis machining and webinar goals
4:30	Geometry Requirements for 3-Axis	Solids, surfaces, meshes — what works best
9:00	Setup, Stock Definition & Work Zero	Bounding box stock, alignment, work zero placement
14:00	Horizontal Roughing (Demo)	Z-level roughing with step-down, stock to leave, masking surfaces
28:00	Parallel Finishing (Demo)	X/Y direction passes, step-over, tolerance, containment
38:00	Horizontal Finishing (Demo)	Steep area finishing and when to use it
47:00	Cut Levels, Stock Allowance & Tool Selection	Flat vs ball mills, roughing vs finishing strategy

Time	Topic / Section	Description
54:00	Simulation, Post-Processing & Resources	Cut material simulation and G-code output
1:02:00	Q&A and Best Practices Wrap-Up	Common questions and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Flat End Mills for Roughing, Ball Mills for Finishing** — Flat end mills calculate faster and remove material more efficiently. Use ball mills (or corner radius mills) only for finishing passes where you need smooth, contoured surfaces.
2. **Leave Stock on Parting Planes** — Never machine all the way to the parting plane with a flat end mill during roughing. Leave 0.010"–0.025" stock and finish later with a ball mill to maintain tangent contact and protect the parting line.
3. **Use Masking Surfaces to Protect Features** — Create simple planar or revolved masking surfaces to prevent the tool from entering holes, pockets, or areas you want to leave stock on during roughing.
4. **Run Parallel Finishing in Two Directions** — Perform one pass at 0° and a second at 90° for dramatically better surface finish on contoured areas.
5. **Step Minimization (if available)** — Combine roughing and re-roughing into one operation by enabling step minimization. This is far more efficient than two separate operations.
6. **Set Reasonable Tolerances** — Use 0.005"–0.01" for roughing and tighten to 0.001" only for final finishing. Tighter tolerances dramatically increase calculation time.
7. **Work Zero at Top of Stock** — Set your work zero at the highest point of the stock for intuitive depth management.
8. **Always Simulate Before Posting** — Run full cut material simulation with fine accuracy after every major operation, especially before drilling or finishing.
9. **Offset Containment Curves Outward** — When finishing near edges or fillets, offset your containment curve outward by tool radius + clearance so the tool can reach the full tangent line.
10. **Part Files Available** — Download the demonstration mold core part from the link in the video description to follow along.

Key Takeaways

This webinar is the **perfect starting point** for anyone new to 3-axis machining in MecSoft CAM. It clearly explains:

- The live demos on a real mold core make the concepts easy to understand and apply immediately.
- Why 3-axis is different from 2.5-axis (full surface control vs curve control).
- The standard high-quality workflow: Rough → Parallel Finish → Horizontal Finish → Cleanup.
- How to protect critical features like parting planes.
- Prepare clean and organized geometry before programming.
- Match cutting tools carefully to surface geometry and machining goals.
- Optimize stepover settings to balance finish quality and machining time.
- Use machining boundaries to improve toolpath control and efficiency.
- Verify machining operations using simulation before posting code.
- Organize setups and operations carefully for smoother workflows.

4.41 CAM in Education

Link: https://www.youtube.com/watch?v=Ok3_OGkGkE0

This AMS webinar focuses on the role of CAD/CAM technology in education and how RhinoCAM and VisualCAD/CAM are used in high schools, colleges, universities, maker spaces, and technical training programs. The presentation explores how educators introduce CNC machining, CAD modeling, and manufacturing concepts to students through practical hands-on learning projects. Topics include curriculum development, student projects, CNC workflow fundamentals, classroom setup, educational licensing, machining demonstrations, and training resources. Throughout the webinar, the presenters also share practical teaching tips for helping students learn CAD/CAM concepts more effectively while preparing them for careers in manufacturing, engineering, and product design.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview of educational use cases
2:39	MecSoft Corporation Overview & Product Lineup	VisualCAD/CAM, RhinoCAM, and educational licensing options
9:44	Case Study: Sisters High School	Woodworking/engineering program teaching ukulele and guitar building
22:27	Case Study: IYRS School of Technology & Trades	Post-secondary boat building, restoration, and composites program
34:35	Live Demo: VisualCAD/CAM Toolpaths (Sisters HS)	Optimized XY Machining and Adaptive Roughing on guitar parts
40:33	Live Demo: RhinoCAM Toolpaths (IYRS Helmet Mold)	2.5-axis facing, Z-level roughing, and parallel finishing on a composite mold
47:34	Educational Licensing, Support & Pricing	Lab licenses, Campus CAM subscriptions, and educator resources
58:00	Conclusion & Q&A	Final thoughts and closing remarks

Key Tips & Best Practices Extracted

Here are the most valuable takeaways from the webinar for educators and schools:

1. **Ease of Use is Critical for High School Students** — Sisters High School chose VisualCAD/CAM because it is significantly easier for high school students to learn compared to more complex industrial CAM packages. The simpler interface allows students to focus on design and manufacturing concepts rather than fighting the software.
2. **Optimized XY Machining Reduces Cusping** — This feature (shown in the Sisters High School demo) combines Z-level and parallel strategies in one operation, dramatically reducing stair-stepping and improving surface finish on contoured parts like guitar bodies.
3. **Adaptive Roughing Clears Material Efficiently** — The webinar highlighted Adaptive Roughing as a powerful tool for quickly removing large amounts of material in a single operation — ideal for educational projects where students want fast results without complex parameter tuning.
4. **Rhino + RhinoCAM is Ideal for Industry-Aligned Programs** — IYRS uses RhinoCAM because Rhino is already the standard CAD platform in the boat-building and marine industry. Students learn both design and manufacturing in the same environment they will use professionally.
5. **Real Student Projects Increase Engagement** — Both schools emphasized that hands-on projects (ukuleles, guitars, and composite molds) dramatically increase student motivation and retention compared to abstract exercises.
6. **Educational Licensing Options** — MecSoft offers flexible educational licensing, including lab packs and Campus CAM subscriptions. These provide access to full versions at significantly reduced cost for schools.
7. **Strong Support for Educators** — Annual maintenance subscriptions include access to new releases, video tutorials, webinars, and direct technical support — very helpful for instructors who are not full-time CAM experts.
8. **Seamless CAD/CAM Integration Matters** — When CAD and CAM are tightly integrated (as in RhinoCAM), students spend less time on data translation and more time learning actual machining strategies.

Key Takeaways

This webinar is an excellent resource for educators considering adding CAD/CAM to their programs. It proves that MecSoft's software is accessible enough for high school students while still powerful enough for post-secondary technical training.

Biggest lessons:

- Choose software that matches your students' skill level (VisualCAD/CAM for high school, RhinoCAM for industry-aligned programs).
- Real, tangible student projects (musical instruments, molds, boat parts) create high engagement.

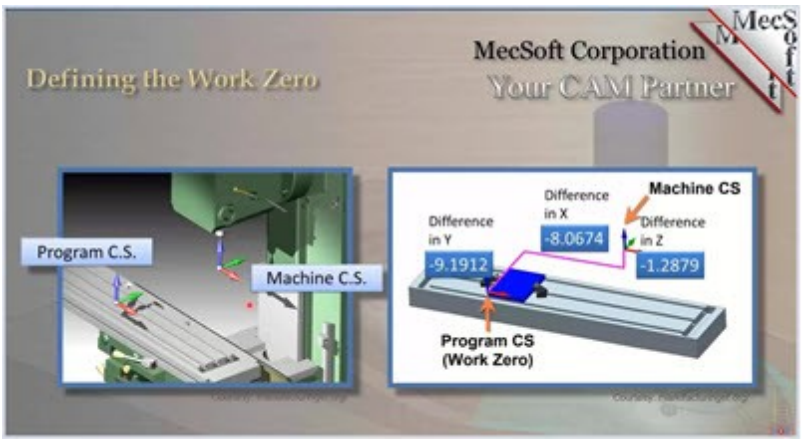
- Features like Optimized XY Machining and Adaptive Roughing deliver professional results with less complexity.
- The webinar also includes a link to a student project video from IYRS (the Mayflower II restoration work), which is worth watching.
- Start students with simple and achievable CNC machining projects.
- Use standardized workflows and project templates for classroom consistency.
- Combine CAD modeling instruction with hands-on machining experiences.
- Use simulation tools to improve safety and machining understanding.
- Introduce students gradually to 2½ Axis and 3 Axis machining concepts.
- Leverage educational resources, tutorials, and CAMJam materials for training.

4.42 2.5 Axis Power User

Link: https://www.youtube.com/watch?v=gbDkP5tMzPw

This AMS webinar focuses on advanced “power user” workflows for 2½ Axis milling in RhinoCAM and VisualCAD/CAM. The presentation explores practical machining strategies used by experienced CAM programmers to improve machining efficiency, reduce cycle time, optimize cut quality, and streamline production workflows. Topics include advanced pocketing, profiling optimization, cut-level management, linking moves, hole machining, control geometry, machining templates, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips designed to help users work faster and create more reliable CNC machining operations for real-world prismatic parts and production machining environments.

Source Files: 20.20.08 AMS Webinar Power User Topics for 2½ Axis Milling.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction and Housekeeping	Webinar goals and overview of power topics
4:24	Tolerances and Stock Parameters	How tolerance affects chord segments, arc fitting, and toolpath quality
18:19	Defining Work Zero	Work coordinate systems, G54/G55, and regeneration requirements
25:51	Programming for Cutter Compensation	G41/G42/G40 usage, linear entry/exit requirements, climb vs conventional
32:55	Engraving and its Many Uses	Text, curves, 3D surfaces, and tool tip behavior
41:06	Using Start Points	Controlling entry locations in profiling and pocketing
49:32	Controlling Cutter Entry/Exits	Linear, radial, helical, long-path, and overlap settings
54:52	Bridges/Tabs in Profiling	Holding nested parts with triangular or rectangular tabs
1:01:15	Utilizing User Defined Tools	Creating custom form tools from half-profile geometry
1:04:16	Fixture Offset Programming	Automating G54/G55 output for multiple parts or fixtures

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Tolerance Directly Controls G-code Size and Finish Quality** — Tighter tolerances create more chord segments and larger files. Use 0.005"–0.01" for roughing and only tighten to 0.001" for final finishing. Always enable Arc Fitting (set to approximately 2× your global tolerance) to convert linear moves into G2/G3 arcs for smoother motion and smaller programs.

2. **Stock-to-Leave Can Be Positive, Negative, or Zero** — Use negative stock in pocketing to push the tool past islands or walls for clean edges. Use zero stock when you want the tool to cut exactly to the geometry.
3. **Work Zero Should Be Set at the Top of Stock** — This makes depth values intuitive. Changing the Work Zero location requires regenerating toolpaths because the coordinate system shifts.
4. **Cutter Compensation Requires Linear Entry/Exit** — G41/G42 only works reliably when the tool has a straight-line entry and exit move. Ramps or arcs during engagement often cause the controller to ignore compensation.
5. **Engraving Drives the Tool Tip** — Unlike profiling, engraving follows the curve with the tool tip (no radius compensation). This makes it ideal for text, V-carving, and following 3D surface curves.
6. **Start Points Give You Precise Entry Control** — In profiling, the curve's start point determines where the tool enters. In pocketing, you can select a pre-drilled hole as the start point for safe, controlled entry.
7. **Long-Path Entry/Exit with Overlap Eliminates Witness Marks** — Use long-path entries with a small overlap distance (e.g., 0.05"–0.10") so the tool overlaps its previous path and removes any visible mark at the entry/exit point.
8. **Bridges/Tabs Keep Nested Parts Attached** — Use triangular or rectangular tabs in profiling operations when cutting multiple parts from a single sheet. Predefined regions let you manually control exact tab locations.
9. **User-Defined Tools for Custom Profiles** — Model only half the tool profile in Rhino/VisualCAD, then load it as a user-defined tool. This is extremely useful for custom form tools, dovetails, or special engraving bits.
10. **Fixture Offset Programming Automates Multi-Part Setups** — Instead of manually editing G-code, use Fixture Offset Programming to automatically output G54, G55, G56, etc., for multiple parts or fixtures on the machine table.

Key Takeaways

This webinar is packed with high-value power techniques that give users fine-grained control over 2.5-axis toolpaths. The biggest productivity and quality improvements come from:

- These techniques remain highly relevant and are still widely used in current versions of RhinoCAM and VisualCAD/CAM.
- Proper tolerance + arc fitting settings
- Strategic use of stock-to-leave (especially negative values)
- Controlling entry/exit behavior and start points
- Using bridges/tabs and fixture offsets for production work

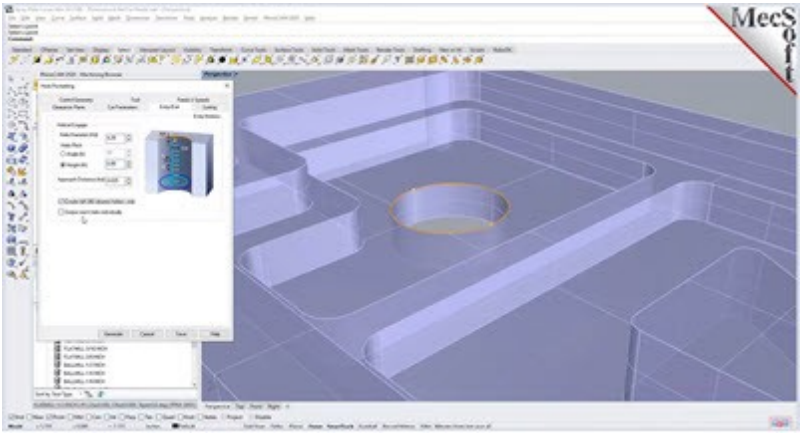
- Creating custom user-defined tools when standard tools aren't enough
- Organize machining geometry by operation type for faster workflow management.
- Optimize linking moves and entry strategies to reduce cycle time.
- Balance cut depths and step-downs for improved cutter stability and tool life.
- Use control geometry to limit unnecessary machine movement.
- Create reusable machining templates to speed up repetitive programming.
- Verify machining operations with simulation before posting NC code.

4.43 2.5 Axis Advanced

Link: https://www.youtube.com/watch?v=Yv8vrsolEYE

This AMS webinar focuses on advanced 2½ Axis machining workflows in RhinoCAM and VisualCAD/CAM. The presentation explores practical machining techniques used to optimize prismatic part machining, improve cutting efficiency, and streamline CAM programming workflows. Topics include advanced pocketing strategies, profiling optimization, cut-level control, drilling workflows, linking moves, machining regions, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for reducing cycle times, improving cutter life, minimizing air cutting, and creating more efficient CNC machining workflows for real-world manufacturing applications.

Source Files: 20.20.07 Webinar Advanced 2 Axis Machining.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview

Time	Topic / Section	Description
2:33	Overview of Topics	Stepped pockets, islands, open pockets, slots, holes, chamfering
3:30	Stepped Pockets with Islands	Pocketing with “Clear Island Tops” + Cleanup Pass
19:32	Open Pockets	Why Facing is better than Pocketing for open regions
24:12	Slotting	Tool matching slot width vs trichoidal clearing
29:50	Whole Pocketing & Whole Profiling (Holes)	Helical entry + spiral cleanup for holes
38:52	Profiling for Finishing	Removing stock left by roughing + Cutter Compensation
43:12	Chamfering (Modeled & Un modeled)	Using V-mills on modeled and un modeled chamfers
47:52	Part Silhouette Profiling	Final outer profile using bottom edges
49:46	VisualCAM Demo + Q&A	Drilling, peck cycles, and linearized output

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **“Clear Island Tops” is Extremely Powerful** — When pocketing stepped areas with islands, enable **Clear Island Tops**. The software automatically generates the correct Z-levels to machine the top of the island without manual intervention.
2. **Use Facing (Not Pocketing) for Open Pockets** — Pocketing assumes a closed region and forces a plunge/ramp. For open pockets, use the **Facing** operation so the tool can enter cleanly from the open side.
3. **Slotting Operation is Smart** — If your tool diameter exactly matches the slot width, the software creates a single center line pass. If the tool is smaller, it automatically switches to a **trichoidal** (circular) clearing pattern.
4. **Whole Pocketing vs Whole Profiling for Holes**:

- Use **Whole Pocketing** when the hole needs to be cleared from solid (includes helical entry + spiral cleanup).
- Use **Whole Profiling** when the hole is already pre-drilled or cleared and you only need to finish the wall.

5. **"Determine Using 3D Model" for Cutting Side** — In profiling operations, let the software automatically decide whether to cut inside or outside the curve based on the 3D model. This is much more reliable than manual selection.
6. **Cutter Compensation Requires Linear Entry/Exit** — If you want to use G41/G42 in your G-code, you must have a straight-line entry and exit move. Ramps or arcs during engagement often cause the controller to ignore compensation.
7. **Chamfering Works on Un modeled Edges** — You don't need to model the chamfer in CAD. Select the top edge, set "Location of Cut Geometry = At Top", and define the chamfer width/height + tool angle. The software calculates the correct toolpath.
8. **Helical Entry + Cleanup Pass for Clean Holes** — Whole Pocketing uses a helical ramp entry followed by a spiral cut and a final cleanup pass at the bottom — excellent for producing accurate, burr-free holes.
9. **Copy Operations to Save Time** — When creating similar operations at different Z-levels, copy the previous operation and simply update the geometry selection and cut level settings.

Key Takeaways

This webinar is excellent for users who want to move beyond basic pocketing and profiling. It focuses on smart use of existing operations (Facing for open pockets, Slotting for slots, Hole Pocketing/Profiling for holes) rather than relying on overly complex strategies.

Biggest lessons:

- The techniques shown remain very relevant and are still widely used in current versions of RhinoCAM and VisualCAD/CAM.
- Choose the right operation for the geometry type (Facing vs Pocketing, Whole Pocketing vs Whole Profiling).
- "Clear Island Tops" and automatic cutting side detection save significant programming time.
- Proper entry/exit control and cutter compensation require deliberate setup.
- Organize machining geometry by operation type for easier workflow management.
- Optimize linking moves and entry strategies to reduce cycle time.
- Balance cut depths and step-downs for improved cutter stability and tool life.
- Use control geometry to minimize unnecessary machine movement.
- Reuse machining templates and workflows to improve programming efficiency.

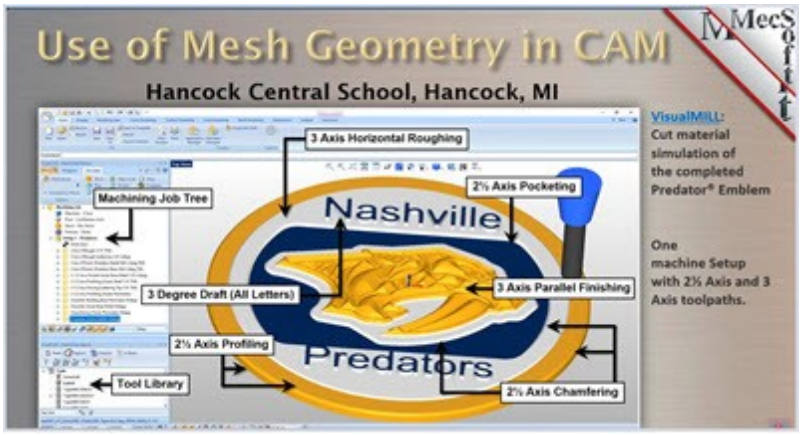
- Verify machining operations using simulation before posting NC code.

4.44 Mesh with CAM & ART

Link: https://www.youtube.com/watch?v=3eJkDZSQBrk

This AMS webinar focuses on using mesh geometry in RhinoCAM and VisualCAD/CAM machining workflows. The presentation explains how mesh-based models such as STL, OBJ, and scanned geometry can be prepared and machined effectively using MecSoft CAM tools. Topics include importing mesh files, mesh cleanup, machining strategies for tessellated geometry, roughing and finishing operations, toolpath optimization, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining quality, reducing calculation issues, and creating efficient CNC workflows when working with scanned data and polygon mesh models.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Overview	Why mesh geometry is important in CAM and what will be covered
3:00	What is Mesh Geometry?	Triangle-based geometry (“triangle soup”), open vs closed meshes, supported file formats (STL, PLY)
7:30	ART Module – Raster to Vector	Converting images (JPEG, TIFF, BMP) into vector curves for relief work
14:00	ART Module – Creating 3D Reliefs	Puff Volume, Swept Volume, and Create 3D Relief from images

Time	Topic / Section	Description
22:00	Machining Artistic Reliefs	Exporting reliefs to CAD and generating toolpaths (Z-level roughing + parallel finishing)
28:00	Mesh Module – Importing & Repairing Meshes	Importing STL/PLY/point clouds, diagnostics, auto-fix, normal correction
35:00	Mesh Optimization	Decimation (reducing triangle count), mesh splitting, and capping
42:00	Machining Imported Meshes	Z-level roughing, 3D Offset / Constant Scallop finishing on mesh geometry
48:00	Real-World Use Cases & Examples	School projects, ZBrush models, 3D scanned parts, and 4-axis indexing on meshes
52:00	Q&A and Best Practices Wrap-Up	Final recommendations and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Meshes Are "Triangle Soup"** — MecSoft CAM can machine open or unconnected triangle meshes (unlike 3D printing, which usually requires watertight closed meshes). This flexibility is very useful for imported scan data or artistic work.
2. **Use the ART Module for Artistic Reliefs** — The ART Module lets you convert 2D images into 3D relief meshes using Puff Volume, Swept Volume, or direct Create 3D Relief from gray scale values. These are excellent for signs, plaques, and decorative work.
3. **Always Repair and Optimize Imported Meshes** — Before machining, use the Mesh Module to:
   - Run diagnostics for open edges and self-intersections
   - Use Auto Fix to repair holes and flip normals
   - Decimate (reduce) high-poly meshes (e.g., from 1.8 million triangles down to a few hundred thousand) to dramatically improve performance
4. **Decimation is Critical for Performance** — Very dense meshes (millions of triangles) significantly slow down toolpath calculation and simulation. Always decimate meshes to a reasonable density while preserving important detail.

5. **Split and Cap Large Meshes** — Use the Mesh Split tool to cut a large mesh along a plane and automatically cap the result. This creates a flat base that is much easier to fixture and machine.
6. **Z-Level Roughing + 3D Offset Finishing Works Well on Mes**hes — For imported mesh models, a typical workflow is:
   - Z-level (Horizontal) Roughing
   - 3D Offset Pocketing or Constant Scallop Finishing for uniform finish
7. **Background Bitmaps as an Alternative** — Instead of using Raster to Vector, you can import an image as a Background Bitmap in VisualCAD/CAM and manually trace curves. This is often more controllable for complex artwork.
8. **Meshes Work with 4-Axis Indexed Machining** — You can split, decimate, and then machine mesh models using indexed 4-axis setups (e.g., rotating the part to machine multiple sides).
9. **Export Reliefs to CAD Before Machining** — After creating a relief in the ART Module, export it as a mesh or polysurface into the main CAD environment before setting up stock and generating toolpaths.
10. **Point Cloud Support** — You can import raw point cloud data (e.g., CSV from 3D scanners) and directly create a mesh from it using the Mesh Module.

Key Takeaways

This webinar is one of the best resources for understanding how to work with mesh geometry in MecSoft CAM. It clearly shows two different but complementary workflows:

- ART Module → For creating artistic reliefs from images.

Biggest lessons:

- Always repair and decimate imported meshes before machining.
- The ART Module is excellent for signage and relief work.
- Standard 3-axis strategies (Z-level roughing + 3D Offset finishing) work very well on optimized meshes.
- Clean and repair mesh geometry before creating toolpaths.
- Reduce excessive mesh density to improve software performance.
- Optimize roughing workflows before refining finishing operations.
- Balance mesh detail and machining tolerances against cycle time.
- Use simulation to validate machining workflows before production.
- Apply proper setup orientation for scanned and tessellated models.

4.45 CAM in High School

Link: https://www.youtube.com/watch?v=IelB_yNmfIU

This AMS webinar focuses on how CAD/CAM technology is being introduced and taught in high school manufacturing, engineering, and STEM education programs using RhinoCAM and VisualCAD/CAM. The presentation explores classroom CNC workflows, beginner machining projects, educational licensing, curriculum integration, and practical methods for teaching students real-world manufacturing skills. Topics include CAD modeling, CAM programming, machine setup, simulation, student projects, and classroom management strategies. Throughout the webinar, the presenters also share practical teaching tips for helping students build confidence with CAD/CAM technology while preparing them for careers in machining, engineering, product design, and advanced manufacturing.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview of educational use cases
3:00	MecSoft Corporation & Product Overview	VisualCAD/CAM, RhinoCAM, and educational licensing options
9:00	Instructor Background & Teaching Philosophy	Doug McIntosh's background and why he teaches manual G-code first
14:00	Student Projects & Curriculum	Sand casting requirement, Bulldog Plaque, and Predators Emblem projects

Time	Topic / Section	Description
20:00	Visual CAD Workflow	Using background bitmaps to trace 2D geometry for logos/emblems
24:00	ART Module – Puff Volume	Converting 2D curves into 3D relief meshes in one step
30:00	MILL Module Demo – Roughing	Horizontal roughing with Step Minimization feature
36:00	MILL Module Demo – Finishing	Spiral and parallel finishing on 3D reliefs
42:00	Educational Licensing & Resources	Lab licenses, Campus CAM, AMS benefits, and teacher support
50:00	Q&A and Wrap-Up	Final questions and closing remarks

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Teach Manual G-Code First** — Doug strongly believes students should manually program simple parts (e.g., cutting a box or metal stamps) before using CAM. This builds deep understanding of Cartesian coordinates, machine movements, and G-code, making the transition to automated toolpaths much more meaningful.
2. **Use Background Bitmaps for Tracing** — When working with complex logos or emblems, import the image as a **Background Bitmap** in Visual CAD. This allows students to accurately trace 2D NURBS curves over the reference image.
3. **Puff Volume = One-Click 3D Reliefs** — The **ART Module's Puff Volume** command is incredibly powerful for education. It converts 2D closed curves into a 3D mesh relief with just one command and dialog box — perfect for student projects like school mascots or emblems.
4. **Step Minimization Saves Time & Improves Finish** — In Horizontal Roughing, enable **Step Minimization**. This automatically adds re-roughing passes in areas with steep steps, resulting in a much smoother in-process stock and protecting smaller finishing tools.
5. **Knowledge Bases for Consistent Teaching** — Teachers can save complete machining strategies (tools, tolerances, step-over, entry/exit, etc.) as **Knowledge Bases**. Students can then load the correct Knowledge Base and only need to select geometry — greatly reducing setup errors and teaching time.

6. **Spiral Finishing for Smooth Reliefs** — For raised 3D reliefs (like the Bulldog plaque), **Spiral Machining** creates a continuous toolpath with no retracts, producing a very smooth surface finish with minimal handwork.
7. **Real Projects Increase Engagement** — Requiring students to design and machine a sand-casting pattern (positive) that will actually be used to cast aluminum parts dramatically increases motivation and gives students a tangible, real-world outcome.
8. **Educational Licensing is Flexible** — MecSoft offers **Lab Licenses** (starting at 5 users + instructor key) and **Campus-Wide Licenses**. Annual Maintenance Subscriptions (AMS) include technical support, CAMJam video tutorials, and monthly webinars — very helpful for teachers who are not full-time CAM experts.
9. **Proof-of-Concept Machining** — Students first machine the design in cherry wood (or similar) as a proof of concept before machining the actual aluminum casting pattern. This teaches the importance of verification and reduces costly mistakes on final material.

Key Takeaways

This webinar is an excellent real-world case study of how **VisualCAD/CAM** (particularly the **ART + MILL** workflow) can be successfully taught in a high school setting.

Biggest lessons:

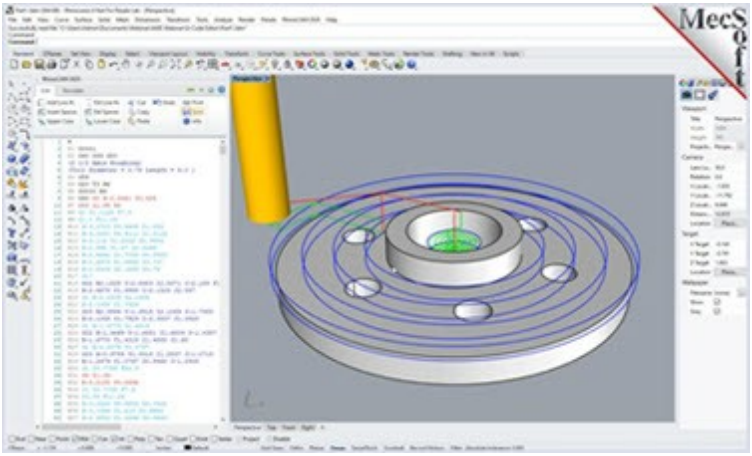
- The session proves that professional-grade CAM software can be taught effectively at the high school level when the curriculum and tools are chosen thoughtfully.
- Start students with simple and achievable machining projects.
- Use standardized training exercises to simplify instruction.
- Teach simulation workflows before students operate CNC machines.
- Connect CAD design concepts directly to physical manufacturing projects.
- Use hands-on machining experiences to improve student engagement.
- Leverage CAMJam and educational resources for classroom instruction.

4.46 G Code Editor

Link: https://www.youtube.com/watch?v=b2DWzxJO9JI

This AMS webinar focuses on the G-Code Editor module in RhinoCAM and VisualCAD/CAM. The presentation explains how users can review, edit, verify, and optimize NC programs directly inside the MecSoft CAM environment. Topics include loading G-code files, syntax highlighting, editing tools, backplot verification, machine simulation, code comparison, search and replace functions, posting workflows, and troubleshooting techniques. Throughout the webinar, the presenters also share practical NC programming tips for improving editing efficiency, reducing machine errors, and streamlining CNC workflow verification before sending programs to the machine.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Why the G-Code Editor Exists	Difference between internal toolpath simulation and actual posted G-code
4:30	Typical Workflow	Program in Mill → Post-process → Verify in G-Code Editor
8:00	Loading G-Code & Tool Crib Setup	Mapping tool numbers from NC file to actual tool definitions
13:00	Simulation Modes	Animation vs Cut Material Simulation
18:00	Editing Capabilities	Text editing, syntax highlighting, merging files
24:00	Instancing on G-Code	XY Instance and Z Instance directly on posted code
29:00	Search & Navigation Tools	Jump to tool changes, feed rates, spindle speeds
33:00	Preferences & Circle Format	Critical settings for accurate back-plotting (Absolute vs Incremental arcs)
38:00	Limitations & Future Plans (as of 2020)	3-axis only, no collision detection yet, planned reverse posting

Time	Topic / Section	Description
42:00	Q&A and Best Practices Wrap-Up	Final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Always Verify the Posted G-Code** — Internal toolpath simulation in the Mill module does not catch errors introduced during post-processing (e.g., wrong arc definitions, incorrect tool changes, or unexpected Z plunges). The G-Code Editor simulates the actual NC file that will run on your machine.
2. **Map Tool Numbers Correctly** — G-code files only contain tool numbers (T1, T2, etc.). You must load the same tool library (CSV) used in the CAM module and map those numbers in the Tool Crib so the simulation knows the actual tool diameters, lengths, and types.
3. **Use Cut Material Simulation When Possible** — Animation mode only shows motion. Cut Material Simulation shows actual stock removal — much more useful for verification.
4. **Merge Multiple NC Files Easily** — You can combine several G-code files into one program. After merging, delete old M30 ends and use the re-sequence line numbers function for clean output.
5. **XY and Z Instancing on G-Code** — You can create multiple copies of a part (XY Instance) or multiple depth passes (Z Instance) directly on the posted G-code without going back to the CAM module. Very useful for production work.
6. **Circle Format Must Match Your Post** — This is the most critical preference setting. If your post outputs Absolute arc centers (I/J/K), set the G-Code Editor to Absolute. If it uses Incremental, set it to Incremental. Mismatch causes completely wrong arc back-plotting.
7. **Syntax Highlighting Improves Readability** — Customize colors for rapid moves (G0), linear moves (G1), and arcs (G2/G3) to make large programs easier to read and debug.
8. **Quick Navigation Shortcuts** — Right-click in the editor to quickly jump to the next tool change, feed rate change, or spindle speed change — extremely helpful when troubleshooting long programs.
9. **Limitations** — At the time of this webinar, the module supported only 2.5-axis and 3-axis milling. 4/5-axis and Turning were not yet supported. Collision detection and reverse-posting were planned for future releases.
10. **Best Practice Workflow:**
    - Program and simulate in the Mill module

- Post-process to G-code
- Open in G-Code Editor
- Load matching tool library
- Run Cut Material Simulation
- Fix any issues found before sending to the machine

Key Takeaways

This webinar clearly demonstrates why the G-Code Editor Module is a valuable AMS-exclusive tool. It bridges the gap between what you see in the CAM software and what actually runs on your CNC machine.

- Use syntax highlighting and search tools to identify problem areas quickly.
- Make incremental edits rather than rewriting entire NC programs.
- Simulate edited code before sending it to the machine.
- Use backplot verification to validate machining motion visually.
- Maintain organized revision workflows for edited NC programs.

Biggest value:

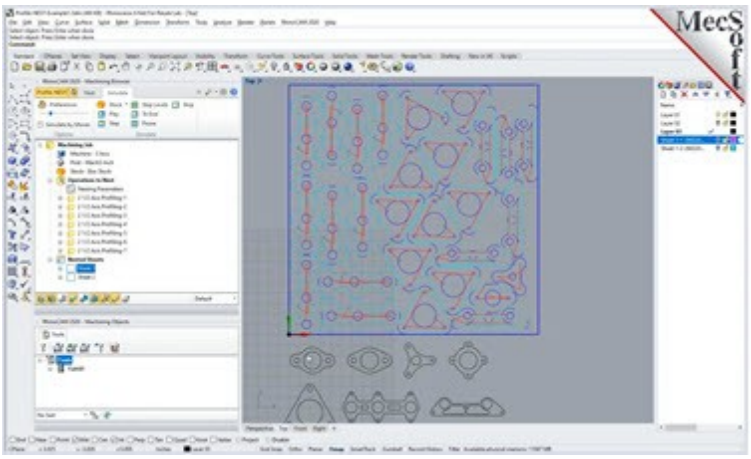
- Catches post-processor translation errors
- Allows editing and instancing directly on G-code
- Provides accurate cut material simulation of the real NC file
- Always review posted G-code before machining production parts.

4.47 Profile-NEST

Link: https://www.youtube.com/watch?v=iYItx9BgE_g

This AMS webinar focuses on profile nesting workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how profile nesting optimizes material usage by automatically arranging parts on sheet stock for CNC cutting operations. Topics include nesting setup, sheet definition, part spacing, grain direction control, common-line cutting, toolpath generation, remnant management, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving material utilization, reducing cutting time, minimizing waste, and streamlining CNC nesting workflows for woodworking, fabrication, plastics, and sheet material manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Module Overview	What Profile NEST is and how it differs from standard nesting
4:00	Profile NEST vs Standard Nesting	Why nesting toolpaths (not just geometry) matters
8:00	Setting Up Profiling Operations for Nesting	Recommended settings for entry/exit, tabs, and cut parameters
14:00	Adding Parts to the Profile Nest Browser	Quantity, priority, fixed orientation, and grain direction
19:00	Sheet Definition & Nesting Parameters	Sheet size, true shape vs rectangular, spacing, remnant control
26:00	Live Demo: Running the Nest	Executing the nest and previewing results
32:00	Editing the Nest & Fine-Tuning	Moving parts, changing orientation, and re-nesting
37:00	Committing the Nest & Layer Management	Committing to separate layers with consistent work zero
41:00	Post-Processing & Best Practices	Posting nested sheets and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Profile NEST Nests Toolpaths, Not Just Geometry** — This is the biggest advantage. It accounts for the full toolpath (including entry/exit moves), making nesting much safer and more accurate than standard geometry nesting.
2. **Use Long-Path or Ramp Entry/Exit** — When setting up profiling operations for nesting, use long-path or ramp entries. This ensures the nesting algorithm sees the full extent of the toolpath and avoids collisions between parts.
3. **Set Realistic Part-to-Part Spacing** — Base spacing on your tool diameter + desired clearance (e.g., tool diameter + 0.100"). This prevents parts from touching after cutting.
4. **Use Priority & Fixed Orientation** — Give critical parts higher priority so they are placed first (bottom-left). Use **Fixed Orientation** when parts must stay in a specific rotation (e.g., for grain direction or aesthetics).
5. **True Shape Nesting is Usually Better** — For irregular or complex shapes, True Shape nesting packs parts much more efficiently than rectangular nesting.
6. **Remnant Control is Powerful** — Set a minimum remnant width so the software doesn't create unusable scraps. You can also label remnants externally for easy warehouse tracking.
7. **Stacked Layout + Separate Layers** — Use stacked layout and commit each sheet to its own layer. This keeps one consistent work zero for the entire job, making downstream CNC programming much easier.
8. **Break Complex Parts into Separate Operations** — For parts with floating segments or complex shapes, create separate profiling operations. This gives you more control during editing and re-nesting.
9. **Knowledge Bases for Repeatability** — Save your profiling operation settings (tolerances, step-over, entry/exit, tabs, etc.) as a Knowledge Base so you can quickly apply the same parameters to new parts.
10. **Edit Individual Operations After Nesting** — You can edit specific operations (e.g., change entry/exit on one part) without re-running the entire nest.
11. **Post Using Standard Mill Posts** — Profile NEST uses the same post-processors as the Mill module, so it works with routers, plasma, waterjet, etc.

Key Takeaways

This webinar clearly explains why the **Profile NEST module** is a valuable AMS-exclusive tool:

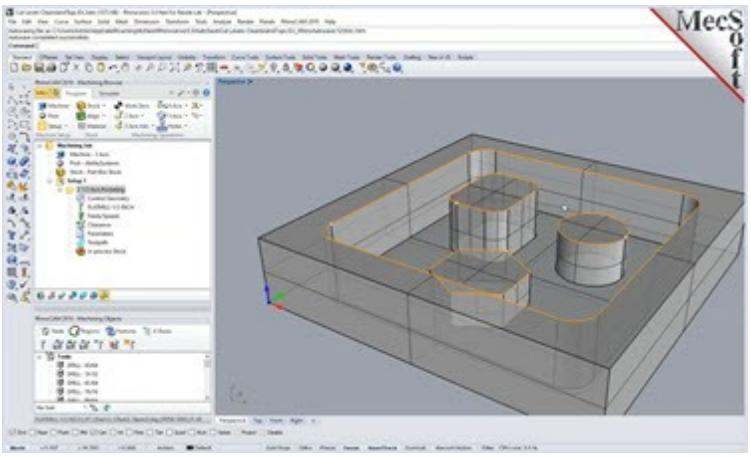
- Best for: Shops doing production nesting of 2.5-axis profiled parts (signs, brackets, enclosures, decorative panels, etc.) who want better material utilization and fewer collisions.
- It gives you more control and safety by nesting actual toolpaths instead of just outlines.
- It integrates seamlessly with 2.5-axis profiling operations.
- Editing and re-nesting is flexible and efficient.
- Organize and clean geometry before generating nested layouts.
- Optimize part spacing to balance material usage and machining safety.
- Use automatic nesting first, then refine layouts manually if needed.
- Apply common-line cutting strategies to reduce waste and cycle time.
- Optimize cut sequencing to improve CNC machine efficiency.
- Reuse leftover material through remnant management workflows.

4.48 Common Support

Link: https://www.youtube.com/watch?v=qEv8mGwe8nw

This AMS webinar focuses on the most commonly requested support topics and troubleshooting questions for RhinoCAM and VisualCAD/CAM users. The presentation covers practical solutions to recurring CAM setup, toolpath generation, simulation, post-processing, geometry selection, and machining workflow issues reported by customers. Topics include geometry cleanup, machining regions, toolpath regeneration, feeds and speeds, post processors, simulation troubleshooting, and workflow optimization. Throughout the webinar, the presenters also share practical CAM programming tips for reducing common mistakes, improving machining reliability, and streamlining CNC programming workflows for everyday production environments.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Why These Topics Matter	Overview of the most frequent support requests
4:30	Location of Cut Geometry	At Top, At Bottom, and Pick Top explained with examples
15:00	Cut Control Levels & Depth Management	Total Cut Depth, Rough/Finish splits, Depth Per Cut, Clear Island Tops
25:00	Part / Stock Alignment	Z-alignment options, XY centering, and "Ignore Wireframe Geometry"
33:00	Work Zero Best Practices	Setting Work Zero correctly for reliable depth control
40:00	Cut Start Side (Inside/Outside & Left/Right)	How curve direction and "Determine Using 3D Model" work
48:00	Step Over Control in Profiling	Total Width + Step Over Cut settings
52:00	Q&A and Common Mistakes Recap	Final tips and closing remarks

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Location of Cut Geometry** is One of the Most Misunderstood Settings:
    - **At Top**: Cut starts from the top of the selected geometry.
    - **At Bottom**: Cut stops at the bottom of the selected geometry (very useful for facing finished surfaces).
    - **Pick Top**: Lets you manually pick a Z point to define the start depth — extremely useful when your control geometry is not at the actual top or bottom you want to machine from.
2. **Use "At Bottom" When Facing Finished Surfaces** — When facing a finished part face, select the face and set Location of Cut Geometry = At Bottom. This prevents the tool from cutting below the final dimension.

3. **Clear Island Tops Saves Huge Time** — In pocketing operations with islands at different heights, enable Clear Island Tops. The software automatically adds passes to clean the top of each island at the correct Z-level without needing separate operations.
4. **Ignore Wireframe Geometry During Stock Alignment** — When working from 2D drawings, always check Ignore Wireframe Geometry during stock alignment. This prevents 2D curves from skewing the stock bounding box.
5. **Set Work Zero at the Top of Stock** — This is the most reliable and intuitive convention. All depth values then become positive numbers going down from zero.
6. **"Determine Using 3D Model" is Usually Best for Closed Curves** — When profiling, let the software automatically decide inside/outside based on the solid model instead of manually choosing "Use Inside" or "Use Outside."
7. **Curve Direction Matters for Open Curves** — For open curves (like slots or open profiles), the software cuts to the left of the curve direction by default. Always check the arrow direction and reverse the curve if needed.
8. **Step Over in Profiling** — When a profile is too wide for one pass, use Total Width + Step Over Cut instead of manually creating multiple operations. You can also set the cut direction to "Mixed" (climb + conventional) or "Climb Only."
9. **Pick Top is Your Friend** — When your control geometry (e.g., a chamfer edge or intermediate curve) is not at the actual machining start depth, use Pick Top to define the correct Z start without moving your CAD geometry.
10. **Stock Alignment Should Usually Be "At Top" or "At Center"** — "At Top" is best when facing from the top. "At Center" is best when you plan to flip the part and machine both sides.

Key Takeaways

This webinar is extremely valuable because it directly tackles the settings that cause the most support calls. The biggest takeaways are:

- These fundamentals remain just as relevant in current versions of RhinoCAM and VisualCAD/CAM.
- Location of Cut Geometry and Work Zero are the two settings that cause the most depth-related errors.
- Clear Island Tops and Determine Using 3D Model dramatically reduce manual work and mistakes.
- Proper stock alignment + ignoring wireframes prevents many downstream problems.
- Clean and organize geometry before generating toolpaths.
- Regenerate operations after major CAD or setup changes.
- Use simulation to verify every machining setup before production.

- Organize operation trees by machining strategy and setup sequence.
- Verify feeds, speeds, and cutter definitions carefully.
- Review post processor output before machining production parts.

4.49 RhinoCAM in Edu

Link: https://www.youtube.com/watch?v=m-QzLrZkKdA

This AMS webinar focuses on how RhinoCAM is used in educational environments to teach CAD/CAM, CNC machining, engineering, and manufacturing workflows. The presentation highlights how high schools, technical schools, maker-spaces, and colleges integrate RhinoCAM into STEM and manufacturing education programs. Topics include CAD/CAM curriculum development, CNC fundamentals, classroom workflows, beginner machining projects, simulation, licensing, and training resources. Throughout the webinar, the presenters also share practical teaching tips for helping students develop real-world manufacturing skills while learning CAD modeling and CNC programming concepts.

Source Files: 20.20.09 Open Webinar RhinoCAM in Education.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview of educational use cases
3:00	MecSoft Product Overview	RhinoCAM capabilities and fit for education
9:44	**IYRS School of Technology & Trades**	Composite technology, boat building & restoration program

Time	Topic / Section	Description
14:00	IYRS Student Capstone Projects	Carbon fiber bicycle helmet mold and “suicide wheel” steering wheel mold
20:00	**Oskayak High School**	STEM program and custom guitar building project
24:00	**RhinoCAM Toolpath Demonstrations**	Facing, roughing, finishing, pencil tracing, and Between Two Curves operations on student molds
30:00	Educational Licensing Options	Lab licenses, Campus CAM, instructor keys, and AMS benefits
34:00	Q&A and Closing Remarks	Student learning curve, Mayflower II project mention, and final thoughts

Key Tips & Best Practices Extracted

Here are the most valuable takeaways from the webinar for educators and schools:

1. **Real Student Projects Drive Engagement** — Both schools emphasized that giving students meaningful, real-world projects (guitar bodies, carbon fiber helmet molds, steering wheel molds) dramatically increases motivation and learning retention compared to abstract exercises.
2. **Rhino + RhinoCAM is Ideal for Industry-Aligned Programs** — IYRS chose RhinoCAM because Rhino is already the standard design platform in the marine and boat-building industry. Students learn both CAD and CAM in the same environment they will use professionally.
3. **Pencil Tracing & Between Two Curves Are Powerful for Molds** — The live demo showed how **Pencil Tracing** and **Between Two Curves** operations are excellent for cleaning up complex mold geometry (such as the steering wheel mold) where standard parallel finishing falls short.
4. **Split Molds & Parting Lines Require Careful Planning** — When machining split molds (like the bicycle helmet mold), proper setup of parting lines, indexing points, and alignment features in RhinoCAM is critical for successful assembly after machining.
5. **Educational Licensing is Very Affordable** — MecSoft offers flexible options including:
    - Lab licenses (starting at 5 seats + instructor key)
    - Campus-wide solutions (Campus CAM)

- Annual Maintenance Subscription (AMS) that includes technical support, version updates, and access to CAMJam tutorials

6. **AMS Provides Strong Educator Support** — The Annual Maintenance Subscription gives schools access to monthly webinars, recorded CAMJam video tutorials, and direct technical support — very helpful for instructors who are not full-time CAM experts.
7. **Proof-of-Concept Machining is Recommended** — Several educators mentioned having students first machine designs in wood or MDF as a proof of concept before moving to final materials (foam, carbon fiber, etc.).
8. **RhinoCAM Runs on Windows Rhino Only** — At the time of this webinar, RhinoCAM was not available for the Mac version of Rhino (this has since changed in later versions).

Key Takeaways

This webinar is an excellent real-world case study showing that **RhinoCAM** is accessible and powerful enough for both high school STEM programs and post-secondary technical training (boat building & composites).

Biggest lessons:

- The session proves that students can successfully machine complex, real-world parts (carbon fiber molds, custom guitars) when given the right tools and projects.
- Industry-aligned software (Rhino + RhinoCAM) gives students a direct pathway into professional careers.
- Features like Pencil Tracing and Between Two Curves are extremely useful for educational mold and artistic projects.
- Affordable lab and campus licensing combined with strong AMS support makes professional CAM viable for schools.
- Start students with simple and achievable CNC machining projects.
- Use repeatable classroom workflows and project templates.
- Connect CAD design directly to physical manufacturing projects.
- Use simulation tools to teach CNC safety and machining concepts.
- Introduce CAD and CAM concepts gradually to beginner students.
- Leverage educational licensing and training resources to support classroom learning.

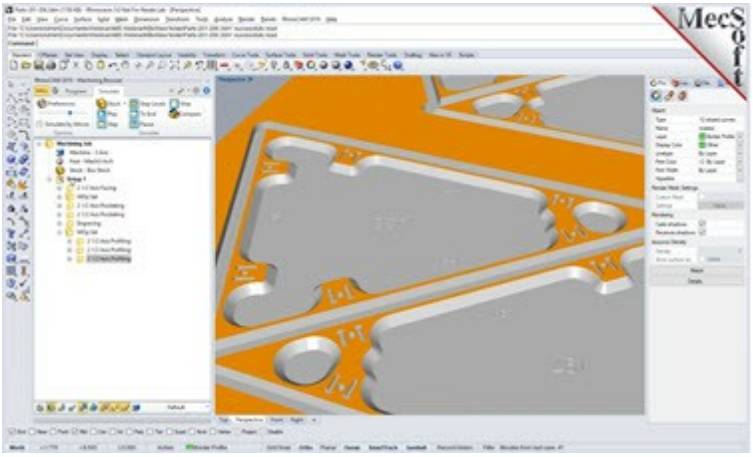
4.50 Knowledge Bases

Link: https://www.youtube.com/watch?v=42e-7LvuNAg

This AMS webinar focuses on using Knowledge Bases (K-Base files) in RhinoCAM and VisualCAD/CAM to automate and accelerate CNC programming workflows. The presentation explains how machining operations, tools, feeds and speeds, machining sequences, and post-

processing settings can be stored and reused across similar parts and production jobs. Topics include creating Knowledge Bases, loading and regenerating operations, machining templates, workflow automation, setup consistency, simulation, and troubleshooting. Throughout the webinar, the presenters also share practical CAM programming tips for reducing repetitive work, improving consistency, minimizing setup errors, and dramatically increasing programming efficiency in production machining environments.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Why Knowledge Bases Matter	Overview of automation and repeatability benefits
4:30	What is a Knowledge Base?	Saving operations, tools, parameters, and selection rules into a reusable .vkb file
9:00	Saving Knowledge Bases	How to save single operations, groups, or entire jobs
14:00	Loading & Applying Knowledge Bases	Drag-and-drop from KB tab, loading tools automatically
20:00	Automation with Selection Rules	Using layer names, colors, or geometry types to auto-assign control geometry
28:00	Feature Machining & Knowledge Bases	Creating reusable workflows for holes (spot drill → drill → profile)

Time	Topic / Section	Description
35:00	Saving Default Parameters	Creating custom defaults for operation types (pocketing, profiling, etc.)
40:00	Multi-Axis & Machine Setup Preservation	Saving machine definitions, tool change points, and post settings
44:00	Q&A and Best Practices	Common questions and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Knowledge Bases Save Entire Workflows** — You can save not just parameters, but complete operations including tools, feeds/speeds, cut levels, entry/exit settings, and even the machine setup (for multi-axis jobs).
2. **Selection Rules = True Automation** — This is the most powerful feature. Create rules based on **layer names**, **colors**, or **geometry types**. When you load the Knowledge Base on a new part with similarly organized geometry, the software automatically selects the correct curves/surfaces and generates toolpaths with zero manual selection.
3. **Feature Machining is Extremely Powerful** — Create a Knowledge Base for common features (e.g., counter-bored holes: spot drill → deep drill → profile). The software can automatically detect matching features on new parts and apply the full workflow.
4. **Load Tools Automatically** — When you load a Knowledge Base, it will automatically add any tools used in the saved operations to your current tool library if they don't already exist.
5. **Save Multiple Default Sets** — Create different Knowledge Bases for different materials (aluminum, steel, wood, plastics) or different machine setups. Then switch your default Knowledge Base in CAM Preferences so new operations start with your preferred settings instead of factory defaults.
6. **Multi-Axis Setups Can Be Saved** — Knowledge Bases can preserve machine definitions, including rotary axis configuration, tool change points, and even the selected post-processor — very useful for shops with multiple machine configurations.
7. **Drag & Drop from the KB Tab** — After loading a Knowledge Base, you don't have to apply everything at once. You can drag individual operations from the KB tab into your Machining Browser exactly where you want them.
8. **Best Practice Workflow**:
    1. Program one good example part completely

2. Organize geometry on consistent layers/colors
3. Save the operations + selection rules as a Knowledge Base
4. On future similar parts, simply load the KB and let the rules do the work

Key Takeaways

This webinar demonstrates one of the **highest-ROI features** in MecSoft CAM. Knowledge Bases (especially when combined with Selection Rules and Feature Machining) can reduce programming time from hours to minutes on families of parts or when repeating similar jobs.

Biggest value highlighted:

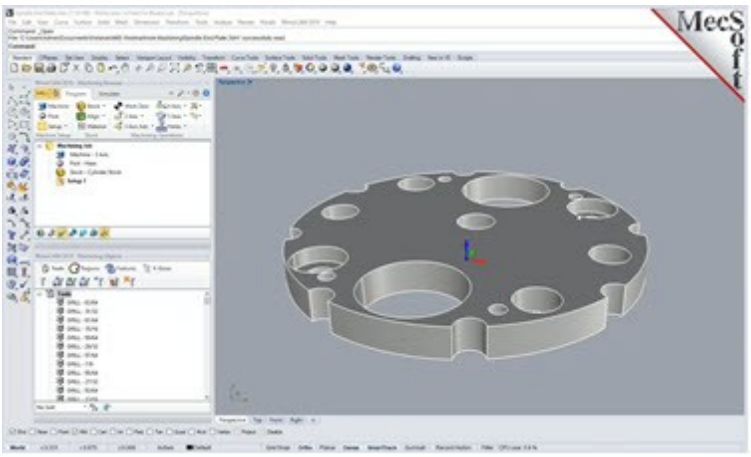
- The techniques shown remain fully relevant and are still one of the best ways to scale productivity in RhinoCAM and VisualCAD/CAM.
- Massive time savings through automation.
- Consistency across jobs and operators.
- Reduced errors from manual geometry selection.
- Easy to create and maintain reusable libraries.
- Build Knowledge Bases around families of similar parts.
- Standardize tooling and machining parameters before saving reusable templates.
- Use Knowledge Bases to eliminate repetitive CAM programming work.
- Verify control geometry carefully when regenerating operations.
- Use drag-and-drop workflows to speed up operation reuse.
- Organize machining templates and K-Base libraries for long-term productivity.

4.51 Hole Machining

Link: https://www.youtube.com/watch?v=RL_GZmlyvrA

This AMS webinar focuses on hole machining workflows in RhinoCAM and VisualCAD/CAM using both 2 Axis and 3 Axis CAM strategies. The presentation explains how drilling, tapping, boring, reaming, hole recognition, and advanced hole machining operations can be programmed efficiently for real-world CNC applications. Topics include geometry selection, automatic hole recognition, drill sequencing, peck drilling, cutter selection, feeds and speeds, 3-axis hole machining strategies, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining efficiency, reducing cycle times, minimizing setup errors, and optimizing hole machining workflows in production environments.

Source Files: 20.19.05 AMS Webinar Hole Machining.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of hole machining topics
3:20	Hole Selection Techniques	2D geometry (points, arcs, circles), 3D solid/surface selection, diameter filtering, flat area selection, predefined regions
19:30	Hole Cycle Operation Parameters	Standard Drill, Peck Drill, Break-Chip Drill, Tapping (standard & peck), countersink, user-defined cycles, dwell, approach, sorting options
26:45	Hole Milling Operations	Hole Profiling (finishing) and Hole Pocketing (larger holes) with helical/spiral entry and cleanup passes
32:50	Feature-Based Machining & Automation	Automatic hole feature detection (through holes, counter-bores, etc.) and generating operations automatically
40:35	Selection Rules for Automation	Using layer names, colors, or geometry types to automatically assign holes to operations
46:40	Output Control & Post-Processing	Canned cycles vs linear motion output, optimized cycle repetition, helical interpolation options, tapping feed rate control
55:00	Conclusion & Q&A	Final recommendations and closing

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Use Diameter Filtering for Fast Selection** — When you have many holes of different sizes, use the Min/Max Diameter filter in the hole selection dialog. This is much faster than manually picking holes one by one.
2. **"Add Tool Tip" for Accurate Hole Depth** — Always enable the "Add tool tip" option when defining drill depth. This ensures the full point of the drill reaches the desired depth (critical for through holes and blind holes that need to be clean at the bottom).
3. **Break-Chip Drilling vs Peck Drilling** — Use Break-Chip for most production work (it breaks the chip without fully retracting). Use Peck Drilling when you need to clear chips completely (deep holes or stringy materials like aluminum).
4. **Sort Holes for Efficiency** — Use Minimum Distance Sorting or directional sorting to reduce unnecessary rapid moves between holes. This can significantly shorten cycle time on parts with many holes.
5. **Predefined Regions + Selection Rules = Automation** — Create named regions for different hole groups (e.g., "M6 Holes", "Counter bores"). Then use Selection Rules in a Knowledge Base so the software automatically selects the correct holes and applies the right operations every time.
6. **Feature-Based Machining Detects Counter bores Automatically** — When using Feature-Based Machining, the software can automatically recognize counter-bores and stepped holes and generate the appropriate spot drill + drill + profile sequence.
7. **Hole Pocketing for Large Holes** — When the hole diameter is significantly larger than your tool, use Hole Pocketing with helical entry instead of trying to drill it. Add a cleanup pass at the final depth for best results.
8. **Optimized Canned Cycle Output** — Enable the option to repeat the canned cycle definition (G81) only once and then output only X/Y coordinates for subsequent holes. This produces much cleaner, shorter G-code.
9. **Helical Interpolation Output** — For controllers that support it, output true helical moves (G02/G03 with I/J) instead of linear segments for smoother, faster hole profiling and pocketing.
10. **Knowledge Bases for Hole Workflows** — Save complete hole-making sequences (spot drill → drill → tap or profile) as a Knowledge Base. On future parts, simply load the KB and let selection rules do the work.

Key Takeaways

This webinar is one of the best resources for mastering efficient and automated hole making in MecSoft CAM. The biggest productivity gains come from:

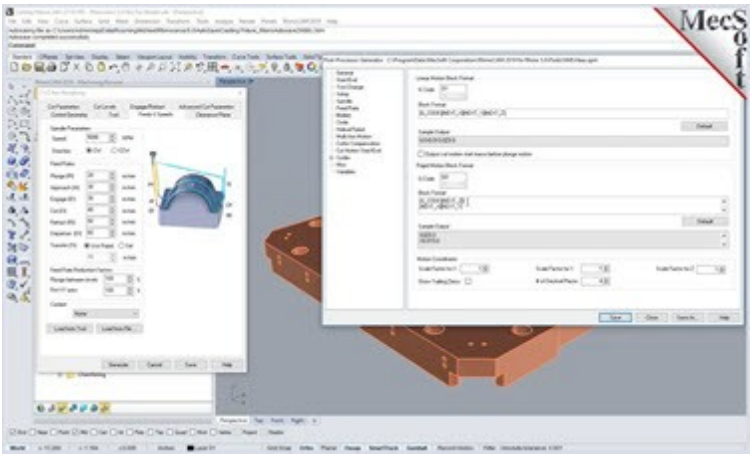
- These techniques remain highly relevant and are still some of the most effective ways to reduce programming time on parts with many holes.
- Smart hole selection (diameter filtering + predefined regions).
- Feature-Based Machining + Selection Rules.
- Proper use of peck vs break-chip cycles.
- Optimized post-processing output (canned cycles + helical interpolation).
- Organize hole geometry before creating machining operations.
- Standardize drill libraries and machining parameters for faster programming.
- Optimize hole sequencing to reduce machine travel and cycle time.
- Verify hole orientation carefully on angled and contoured surfaces.
- Use proper peck drilling strategies for deep-hole machining.
- Validate drilling workflows using simulation before posting NC code.

4.52 Post-Processing Adv2

Link: https://www.youtube.com/watch?v=wAia9TR2vDo

This AMS webinar focuses on post processing and post customization workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how post processors convert CAM toolpaths into machine-specific NC code and how users can customize posts to match CNC machine requirements, controller formats, tooling workflows, and shop standards. Topics include post processor selection, NC output formatting, editing post variables, customizing headers and footers, tool change formatting, simulation, verification, and troubleshooting. Throughout the webinar, the presenters also share practical CAM programming tips for improving machine compatibility, reducing posting errors, and streamlining CNC production workflows.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction and Agenda	Overview of post-processing and customization goals
3:02	Post Processing Workflow Overview	How operations become G-code in RhinoCAM 2019
5:01	Setting Post Processor Options	Selection, file extensions, naming conventions
14:41	Program to Send Posted File To	Custom editors and DNC integration
17:22	Post Processing Multi-Axis Operations	Handling 4-axis and 5-axis indexed setups
23:24	Post Processor Generator (PPG) Overview	Structure and main sections of the PPG
26:37	General Section Settings	Units, comments, sequence numbers
27:29	Tool Change Section	First tool load and subsequent tool changes
29:06	Spindle Section	RPM limits and output control
30:16	Feedrate Section	Units, inverse time, decimal places
31:52	Motion Block Format	Rapid (G0), linear (G1), and scale factors
34:31	Circle Section	Arc formats (IJK vs Radius), helical/spiral handling
38:12	Helical and Spiral Sections	Output options for helical moves
39:37	Multi-Axis Section	4/5-axis output configuration
40:28	Cycles Section	Drill cycle definitions and optimization
42:03	Miscellaneous Section	Coolant, work offsets, and custom output

Time	Topic / Section	Description
42:25	Customizing the Post – Adding Stock Dimensions	Inserting variables as comments
47:07	Using Arithmetic Expressions	e[...] syntax for calculations (e.g., clearance + 1")
51:48	Tips and Tricks	Coolant timing, optimized drill cycles, Machine Control Cycles
58:19	Cut Motion Start/End for Plasma/Laser/Waterjet	Torch/laser on/off control
1:01:58	Conclusion and Q&A	Final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Always "Post All" for Multi-Axis Indexed Jobs** — When you have multiple indexed setups (e.g., 0°, 90°, 180°, 270°), post the entire job together rather than individual setups. This ensures the post correctly outputs rotation angles and work offsets in one clean NC file.
2. **Use "Save As" When Editing Posts** — Never overwrite the original post file. Always use **Save As** so you have a clean fallback if something breaks.
3. **Variables + Arithmetic Expressions are Extremely Powerful** — You can insert variables (e.g., stock dimensions, tool diameter, clearance height) into the post. Use the e[...] syntax to perform calculations, such as automatically adding 1 inch to the clearance height for safety.
4. **Optimized Drill Cycle Output Reduces G-code Size** — Enable the option to output the canned cycle definition (G81, G83, etc.) only once, then output only the X/Y coordinates for subsequent holes. This produces much cleaner, shorter programs.
5. **Coolant Timing Can Be Adjusted** — Move the coolant codes (M8/M9) within the Tool Change section so coolant turns on *before* the first rapid move, not after.
6. **Machine Control Cycles for Custom M-Codes** — Use **Machine Control** operations (or add comments starting with $ in operation properties) to insert specific M-codes (like M01 for optional stop) between operations.
7. **Cut Motion Start/End for Plasma/Laser/Waterjet** — Use the dedicated **Cut Motion Start** and **Cut Motion End** sections in the post to output the correct M-codes to turn the torch/laser on before the cut and off after.

8. **Circle Format Must Match Your Controller** — Choose between IJK (center point) or Radius (R) output depending on what your machine/controller supports. Incorrect format is a common source of alarms.
9. **Test Changes by Regenerating Toolpaths** — Some post-related changes (especially in CAM Preferences) require you to regenerate the toolpath before reposing to see the effect.
10. **Common Mistake: Typos in Variables** — If you make a syntax error in a variable or expression (e.g., missing brackets), the raw macro text will appear in your G-code instead of the calculated value.

Key Takeaways

This webinar is one of the best resources for understanding both **basic post-processing** and **advanced post customization** in MecSoft CAM. The biggest value comes from learning how to use the **Post Processor Generator (PPG)** with variables and arithmetic expressions to create smart, machine-specific posts.

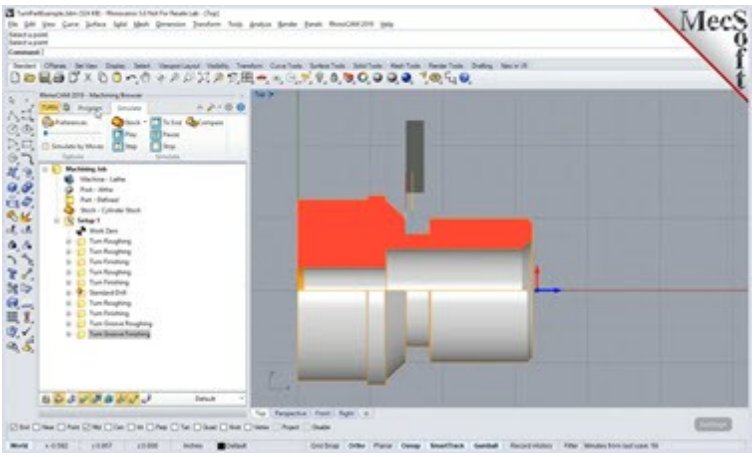
- **Best for**: Users who want to move beyond using default posts and start customizing output for their specific machines (especially for multi-axis, plasma/laser, or highly optimized drill cycles).
- Always match the post processor to the CNC controller being used.
- Review posted NC code before machining production parts.
- Standardize NC formatting across machines and workflows.
- Test customized posts with small programs before full production runs.
- Organize post processors carefully for multi-machine environments.
- Use simulation and backplot tools to verify posted machine motion.

4.53 TURN Basic

Link: https://www.youtube.com/watch?v=bsH5GtMB9VM

This AMS webinar focuses on using MecSoft TURN in RhinoCAM and VisualCAD/CAM to program CNC lathes. The presentation introduces turning workflows used for machining cylindrical parts and explains how lathe programming differs from milling operations. Topics include stock setup, turning tool definitions, facing, rough turning, finish turning, grooving, threading, drilling, simulation, and post processing. Throughout the webinar, the presenters also share practical CNC turning tips for improving machining efficiency, reducing setup errors, optimizing surface finish quality, and streamlining lathe programming workflows for real-world manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the TURN module and what will be covered
4:00	Coordinate System & Part Orientation	Critical setup rules (back face at origin, first quadrant, X/Y swap)
9:30	Stock Geometry Definition	Five methods: Cylinder, Part Cylinder, Revolve, Offset, Stock from Selection
17:00	Tool Definition & Setup	Insert types, orientation (OD Forward/Backward, ID), relief angles, feeds/speeds
24:00	OD Roughing & Finishing	Face roughing, OD roughing with in-process stock, relief angle protection
32:00	ID Operations (Drilling + Turning)	Pre-drilled hole expansion, ID roughing/finishing, shank collision detection
40:00	Grooving Operations	Roughing + finishing passes, program point definition, overlap settings
48:00	Post-Processing & Output Options	Diameter vs Radius mode, work zero, final G-code generation

Time	Topic / Section	Description
55:00	Q&A and Best Practices Wrap-Up	Common questions and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Part Orientation is Critical** — The back face of the part must be at the world origin (0,0,0), and the part must be in the first quadrant (positive X and Y). The TURN module automatically swaps World X → Turn Z and World Y → Turn X. Incorrect orientation causes most setup problems.
2. **Work Zero Should Be on the Rightmost Face** — Set Work Zero to the rightmost face of the stock or part (chuck side). This makes all X coordinates positive and intuitive.
3. **Use "Part Cylinder Stock" for Most Jobs** — This method automatically creates stock based on the part's bounding cylinder with user-defined radial and axial offsets. It is usually the fastest and most accurate method.
4. **Relief Angle Protection is Automatic** — When you define the tool's relief angle correctly, the software automatically prevents the tool from colliding with features like grooves or shoulders during OD and ID operations.
5. **Always Account for In-Process Stock** — The TURN module tracks material removed by previous operations. This prevents the tool from air-cutting and makes roughing much more efficient.
6. **Use Separate Roughing and Finishing Operations for Grooves** — Grooving benefits from a dedicated roughing pass (with step-over and plunge) followed by a finishing pass with overlap. This produces cleaner results than trying to do everything in one pass.
7. **ID Operations Should Start with Drilling** — When machining deep IDs, always drill first (or use a pre-drilled hole). Then use ID roughing/finishing tools. The software can detect shank collisions during ID turning.
8. **Diameter Mode vs Radius Mode** — In CAM Preferences, you can choose whether X output is in Diameter mode (X values are twice the actual radius) or Radius mode. Most lathe controllers expect Diameter mode.
9. **Stock from Selection Works Well for Complex Stock** — If your stock has an irregular shape (pre-machined or cast), use Stock from Selection and select the actual stock solid/surface. In SOLIDWORKS, remember to suppress the stock feature after defining it.
10. **Constant Surface Speed (CSS) is Recommended** — When possible, use CSS instead of constant RPM. This maintains consistent cutting conditions as the diameter changes.

Key Takeaways

This webinar is one of the best practical introductions to MecSoft's TURN module. It clearly explains the critical setup rules (especially part orientation) and demonstrates a complete real-world workflow including OD, ID, and grooving operations.

Biggest lessons:

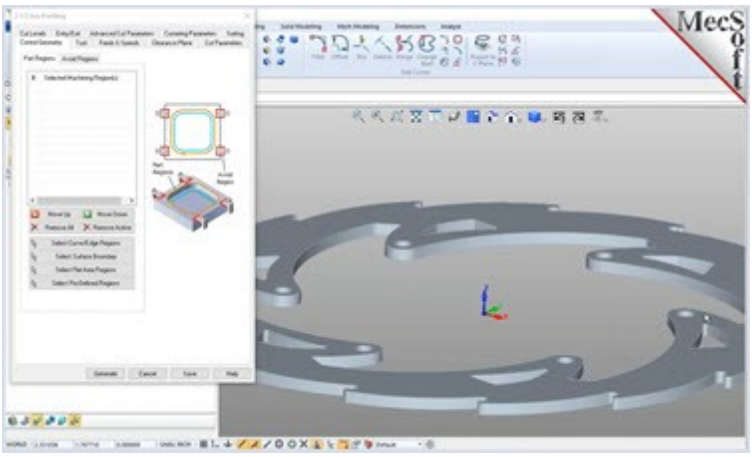
- The techniques shown remain fully applicable in current versions of RhinoCAM and VisualCAD/CAM.
- Proper part orientation and work zero placement prevent the majority of problems.
- The module intelligently tracks in-process stock and tool relief angles.
- Separate roughing + finishing passes for grooves produce the best results.
- Post-processing options (Diameter vs Radius) must match your machine/controller.
- Define stock setup and spindle orientation carefully before programming.
- Standardize turning tool libraries for consistent machining workflows.
- Leave consistent material allowances for finish turning passes.
- Verify threading parameters carefully before posting NC code.
- Organize turning operations logically to improve workflow efficiency.
- Use simulation tools to validate turning motion and setup safety.

4.54 Use of Selections

Link: https://www.youtube.com/watch?v=F9Cy70LFcHo

This AMS webinar focuses on the effective use of selections in machining workflows within RhinoCAM and VisualCAD/CAM. The presentation explains how geometry selection methods directly impact machining accuracy, toolpath generation, workflow efficiency, and operation management. Topics include chain selection, region selection, surface and polysurface selection, selection filters, control geometry, machining boundaries, associative geometry workflows, simulation, and troubleshooting. Throughout the webinar, the presenters also share practical CAM programming tips for improving selection accuracy, reducing programming errors, streamlining operation setup, and creating more efficient CNC machining workflows for both 2½ Axis and 3 Axis machining applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Why effective selections are critical for productivity
3:45	Basic Selection Tools	Single, window, crossing, and chain selection methods
8:20	Regions & Predefined Regions	Creating, naming, and saving reusable geometry groups
15:10	Using Layers and Colors for Selection	Organizing geometry and selecting by layer/color
22:40	Selection Rules in Knowledge Bases	Automating geometry selection based on rules
31:00	Live Demo: Prismatic Part	Full workflow using regions and rules on a plate part
41:20	Live Demo: Complex 3-Axis Part	Using selections for mold cavity and multi-level features
48:50	Best Practices & Q&A	Tips for organizing geometry and scaling automation

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Predefined Regions Are Extremely Powerful** — Create and name regions for common feature groups (e.g., “All 1/4” Holes”, “Pocket Group A”). You can then select them instantly in future operations.
2. **Organize by Layers and Colors** — Assign consistent layer names and colors to geometry types. This allows fast selection and is the foundation for Selection Rules.
3. **Selection Rules + Knowledge Bases = Automation** — Define rules like “All curves on Layer ‘Holes’ = Drilling Operation” inside a Knowledge Base. When you load the KB on a new part, the software automatically selects and machines matching geometry.
4. **Use Chain Selection for Connected Curves** — Hold Shift while clicking to quickly select chains of connected curves — much faster than manual multi-selection.
5. **Save Selections with the Part File** — Predefined regions and selection rules are saved with the part file, making them available every time you open the job.
6. **Combine Selection Methods** — Use a mix of regions, layers, and manual selection for complex parts. Start with broad rules, then refine manually.
7. **Consistent Naming Convention** — Establish a company-wide naming system for layers and regions (e.g., “Holes-M6”, “Pockets-Deep”) to make automation scalable across multiple users.
8. **Test Selection Rules on Simple Parts First** — Before applying rules to production jobs, test them on a simple test part to ensure they behave as expected.

Key Takeaways

This webinar teaches one of the most effective ways to boost productivity in MecSoft CAM: **mastering selections**. When combined with Knowledge Bases and consistent geometry organization, selections turn repetitive manual work into fast, automated processes.

Core message: Well-organized geometry + smart selection rules = dramatically faster and more consistent programming.

- The techniques shown remain fully applicable in current versions of RhinoCAM and VisualCAD/CAM.
- Proper part orientation and work zero placement prevent the majority of problems.
- The module intelligently tracks in-process stock and tool relief angles.
- Separate roughing + finishing passes for grooves produce the best results.
- Post-processing options (Diameter vs Radius) must match your machine/controller.
- Organize CAD geometry carefully before creating machining operations.
- Verify chain direction before generating profile toolpaths.

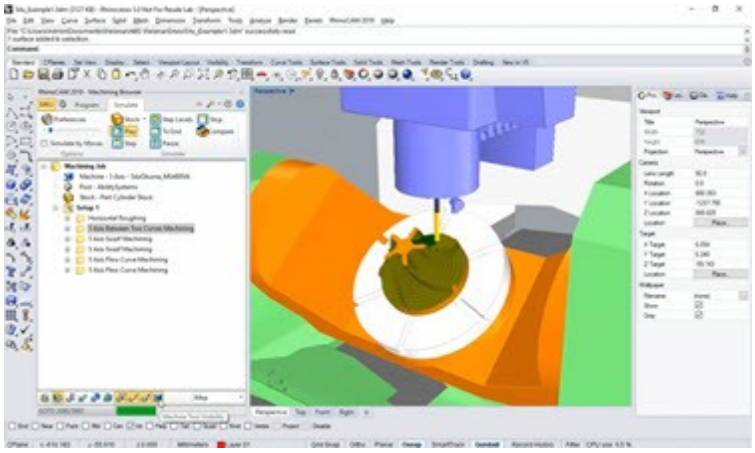
- Use selection filters to simplify complex geometry workflows.
- Keep control geometry separate from machining geometry layers.
- Use machining boundaries to improve toolpath efficiency.
- Verify selections using simulation before posting NC code.

4.55 5 Axis Machining

Link: https://www.youtube.com/watch?v=1WtLuV2sRf4

This AMS webinar focuses on 5-axis machining workflows in RhinoCAM and VisualCAD/CAM. The presentation introduces the fundamentals of simultaneous and indexed 5-axis machining, demonstrating how advanced tool orientation and machine motion allow users to machine complex geometry efficiently and accurately. Topics include machine setup, tool axis control, indexed positioning, simultaneous machining strategies, collision avoidance, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining accessibility, reducing setups, optimizing surface quality, and creating efficient 5-axis machining workflows for real-world CNC applications.

Source Files: 20.21.06 AMS Webinar Understanding 5 Axis Machining.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of 5-axis machining in MecSoft CAM
4:00	5-Axis Concepts	Indexed (3+2) vs Continuous 5-axis explained
9:00	Machine Setup & Configuration	Table/table, head/head, and hybrid 5-axis machine definitions

Time	Topic / Section	Description
15:00	Indexed 5-Axis (3+2) Strategies	Locking rotary axes at fixed angles + standard 3-axis operations
22:00	Continuous 5-Axis Operations	Surface Normal, Curve Projection, Swarf, Flow Curve, and Parallel machining
35:00	Tool Orientation & Collision Avoidance	Automatic and manual tool orientation methods + collision detection settings
45:00	Post-Processing for 5-Axis	Requirements for 5-axis posts, rotary axis output, and work offset handling
52:00	Live Demo & Simulation	Full 5-axis job simulation and G-code review
58:00	Q&A and Best Practices Wrap-Up	Final recommendations and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Start with Indexed (3+2) Whenever Possible** — Indexed 5-axis is generally more rigid, easier to program, easier to verify with simulation, and produces excellent results for most multi-sided parts. Use continuous 5-axis only when the geometry truly requires simultaneous motion of all five axes.
2. **Proper Machine Configuration is Critical** — Define your specific 5-axis machine type (table/table, head/head, or hybrid) accurately in the Machine Tool Setup. Incorrect rotary axis definitions lead to wrong tool orientation and collisions.
3. **Use Automatic Tool Orientation First** — Let the software automatically orient the tool normal to the surface or along a curve before manually adjusting. This is much faster and more reliable for most parts.
4. **Collision Detection Should Always Be Enabled** — Turn on collision detection during toolpath generation and simulation. Set a reasonable clearance value to avoid near-misses that may not show up in basic simulation.
5. **Swarf Machining is Excellent for Ruled Surfaces** — When you have ruled or developed surfaces (common on molds and aerospace parts), Swarf machining keeps the side of the tool in constant contact with the surface — often producing the best finish with fewer passes.

6. **Curve Projection & Flow Curve for Complex Toolpaths** — Use these when you need the tool to follow a specific path or when standard parallel finishing cannot reach all areas cleanly.
7. **Post-Processor Must Fully Support 5-Axis Output** — Not all posts handle simultaneous 5-axis motion correctly. Test thoroughly or request a custom post from MecSoft support. The post must correctly output rotary axis angles (A/B/C) and handle work offset transformations.
8. **Simulate Everything Before Posting** — 5-axis toolpaths are complex. Always run full machine simulation (not just toolpath simulation) to verify there are no collisions between the tool, holder, machine components, and the part/fixture.
9. **Use Indexed for Roughing, Continuous for Finishing** — A common best-practice workflow: rough with indexed 3+2 strategies (more rigid, faster material removal), then finish with continuous 5-axis for superior surface quality.

Key Takeaways

This webinar provides a solid foundation for understanding 5-axis machining in MecSoft CAM. The key message is that 5-axis capability dramatically expands what you can machine, but it also increases complexity — so choose the simplest strategy that gets the job done (indexed first, continuous only when necessary).

Biggest lessons:

- Proper machine setup and tool orientation strategy are the foundation of successful 5-axis work.
- Collision detection and full machine simulation are non-negotiable.
- The post-processor must be correctly configured for reliable 5-axis output.
- Begin with indexed machining before attempting full simultaneous 5-axis workflows.
- Verify tool orientation carefully before generating toolpaths.
- Use tool tilt strategically to improve cutter accessibility.
- Simulate all rotary and tilt-axis motion before machining production parts.
- Monitor machine travel limits and collision zones carefully.
- Organize setups and machining orientations to reduce programming complexity.

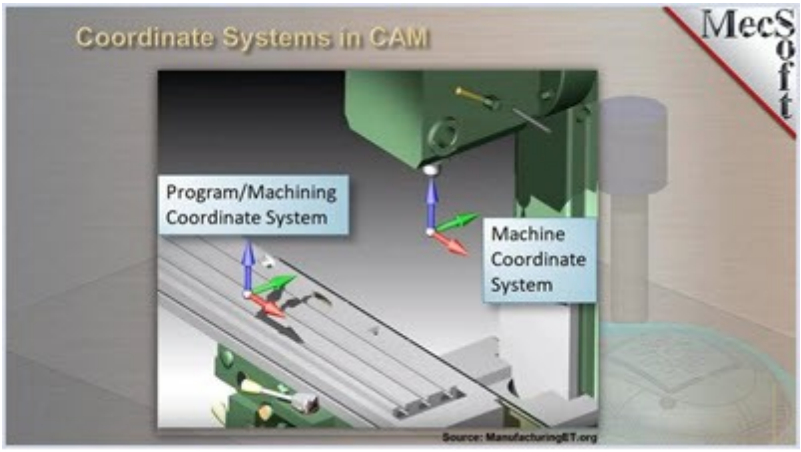
4.56 Multi-Sided Parts

Link: https://www.youtube.com/watch?v=EVASXoxVx0I

This AMS webinar focuses on machining multi-sided parts using 2 Axis and 3 Axis machining workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how users can efficiently machine multiple faces of a part by organizing setups, coordinate systems, and machining

orientations without requiring full simultaneous multi-axis machining. Topics include stock setup, work coordinate systems, indexed setups, part orientation, profiling, pocketing, drilling, 3-axis finishing, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for reducing setups, improving machining accuracy, simplifying multi-sided workflows, and creating efficient CNC machining strategies for real-world production parts.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of multi-sided machining challenges and solutions
4:00	What is 3+2 (Indexed) Machining?	Definition and benefits compared to continuous 5-axis
9:00	Coordinate Systems Explained	Machine Coordinate System vs Machining/Program Coordinate System (G54/G55)
14:00	Multi-Sided Workflows Without Multiple Setups	Creating separate files per side (Standard configuration)
20:00	Multi-Sided Workflows With Multiple Setups	Using Setups in Professional/Premium configurations (recommended method)
26:00	4-Axis Indexed Machining Demo	Using “Rotate Table Setup” command for rotary table machines

Time	Topic / Section	Description
35:00	5-Axis Indexed (3+2) Machining	Head-Head vs Table-Table configurations and Gage Length
42:00	Post-Processor Requirements	Outputting rotary axis angles (A/B/C), safe retracts, and axis limits
48:00	Simulation Across Multiple Setups	Benefits of single-file simulation and machine tool simulation
55:00	Q&A and Best Practices Wrap-Up	Final recommendations and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Use Multiple Setups Whenever Possible** — In Professional or Premium configurations, create one Setup per side/orientation inside a **single part file**. This is far superior to creating separate files for each side because it keeps everything organized, allows full simulation of the complete job, and reduces the risk of alignment errors.
2. **"Rotate Table Setup" is the Key Command for 4-Axis** — For machines with a rotary table, use the **Rotate Table Setup** command. You define the rotation center and angle (e.g., 90°, 180°), and the software automatically transforms the toolpaths and coordinate system for the new orientation.
3. **Gage Length is Critical for Head-Head 5-Axis Machines** — When using a head-head configuration, you must accurately define the **Gage Length** (distance from the rotary pivot point to the tool tip). Incorrect Gage length causes the tool to be in the wrong position after rotation.
4. **Always Retract to a Safe Height Before Rotating** — Configure your post-processor (or use Machine Control operations) to move the tool to a safe clearance height **before** any rotary axis movement. This prevents collisions during indexing.
5. **Post-Processor Must Support Rotary Output** — Your post must be configured to output the correct rotary axis commands (A, B, or C) and angles. You can also set axis limits in the post to prevent the software from calculating angles outside your machine's physical range.
6. **Simulation is Much More Powerful in a Single File** — When all sides are programmed in one file using multiple setups, you can simulate the entire job (including part rotation on 4-axis machines) in one go. This makes collision detection and verification significantly more reliable.

7. **Work Offsets (G54, G55, etc.) for Multi-Part Setups** — For tombstone or multiple-part fixtures, use different work offsets for each indexed position. The post can automatically output the correct G54/G55/etc. commands.
8. **Manual File-per-Side Workflow is Still Valid** — For users on Standard configuration (or simpler 3-axis machines), creating separate files per side is still a valid approach — just more time-consuming and error-prone than using multiple setups.
9. **Machine Tool Simulation Helps Visualize Rotation** — When available (Expert/Pro/Premium), use Machine Tool Simulation to visually see how the part or head will rotate between setups. This is excellent for training and verification.

Key Takeaways

This webinar clearly explains the evolution of multi-sided machining workflows in MecSoft CAM:

- The session remains highly relevant for anyone machining parts that require features on multiple faces using indexed (3+2) strategies on 4-axis or 5-axis machines.
- Older approach: Separate files per side (manual and error-prone)
- Modern recommended approach: Multiple Setups in a single file + Rotate Table Setup command.
- Plan all machining orientations before generating toolpaths.
- Keep machining coordinate systems organized and clearly labeled.
- Group operations logically by setup orientation.
- Verify tool accessibility and holder clearance early in the workflow.
- Use indexed setups to reduce fixture changes and improve accuracy.
- Validate all setups using simulation before posting NC code.

Biggest advantages of the modern method:

- All operations in one file
- Easier simulation and collision checking
- Automatic coordinate transformation
- Cleaner post-processing output

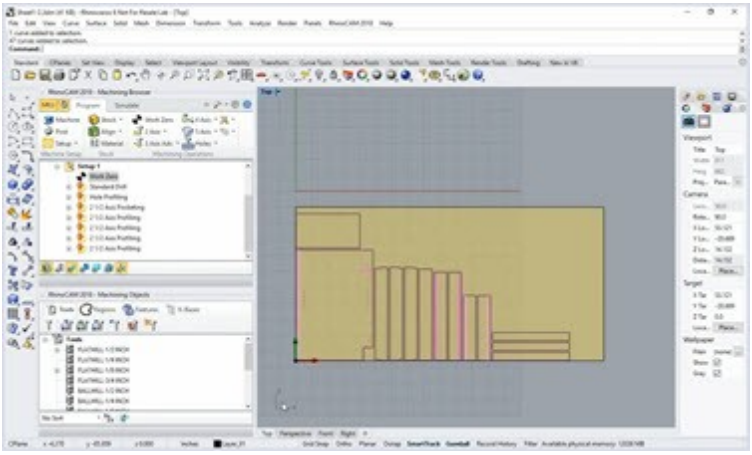
4.57 Parts Nesting

Link: https://www.youtube.com/watch?v=B1Tg8BYmtk0

This AMS webinar focuses on parts nesting workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how nesting technology optimizes sheet material usage by automatically arranging parts for CNC cutting operations. Topics include sheet definition, true-shape nesting,

rectangular nesting, part spacing, grain direction control, nesting constraints, toolpath generation, remnant management, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving material utilization, reducing machining time, minimizing waste, and streamlining nesting workflows for woodworking, plastics, fabrication, and sheet material manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Module Overview	What the NEST module does and why it's useful
4:30	Rectangular Nesting (Box Nesting)	How it works, best use cases, and workflow
12:00	True Shape Nesting	Core concept and advantages over rectangular nesting
18:00	True Shape Advanced Options	Orientation step angle, grain direction, mirroring, parts inside parts
25:00	Non-Rectangular Sheets & Remnants	Nesting on irregular or remnant stock
30:00	Tagging & Reporting	Automatic part labeling and nest reports
35:00	Workflow: Estimate → Execute → Commit	Full nesting process from start to finish

Time	Topic / Section	Description
42:00	Integrating Nests with the Mill Module	Moving nested sheets into machining operations
48:00	Using Knowledge Bases & Selection Rules	Automating toolpath generation on nested parts
54:00	Q&A and Best Practices	Final tips and closing remarks

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **True Shape Nesting is Almost Always Better** — It uses the actual geometry of the parts instead of bounding boxes, resulting in significantly better material utilization, especially for irregular or organic shapes.
2. **Use "Parts Inside Parts" When Possible** — This powerful option allows smaller parts to be nested inside the holes or interior cutouts of larger parts. It can dramatically increase sheet efficiency.
3. **Grain Direction & Fixed Orientation** — When working with wood, composites, or materials with directional properties, use Grain Direction and Fixed Orientation to prevent parts from rotating in ways that would violate material requirements.
4. **Orientation Step Angle** — Instead of allowing free rotation, set a step angle (e.g., 15° or 90°) to control how much parts are allowed to rotate. This often produces more practical and machinable nests.
5. **Always Estimate Sheets First** — Before running the nest, use the Estimate Number of Sheets feature. This saves time by telling you how many sheets you'll need without committing to a full nest calculation.
6. **Non-Rectangular / Remnant Sheets** — True Shape nesting supports irregular sheet shapes. You can import remnant sheets and nest directly onto them — a major advantage over rectangular nesting.
7. **Tagging Parts Automatically** — Use the tagging feature to automatically add text or curve-based labels to parts during nesting. This makes it much easier to identify parts after cutting.
8. **Knowledge Bases + Selection Rules = Automation** — After committing a nest, use Knowledge Bases combined with Selection Rules (based on layer, color, or geometry type) to automatically apply machining operations (drilling, pocketing, profiling) to all nested parts with almost no manual work.

9. **Commit to Separate Layers** — When committing the nest, choose to put each sheet on its own layer. This keeps everything organized and makes it easy to machine one sheet at a time while maintaining a consistent work zero.
10. **Sign Making Tip** — For text and complex signs, group interior and exterior curves together and enable the "Use for engraving and sign making" option so the entire sign is treated as one part during nesting.

Key Takeaways

This webinar clearly demonstrates why the NEST module (especially True Shape nesting) is a powerful tool for improving material efficiency and reducing waste.

Biggest advantages highlighted:

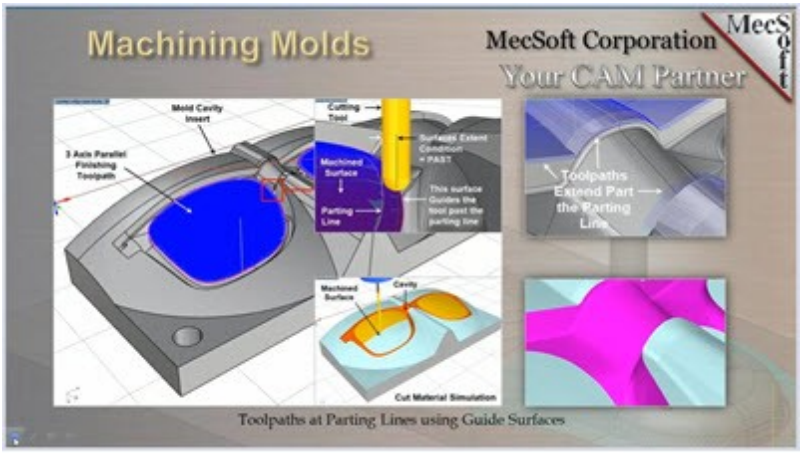
- The techniques shown webinar remain very relevant and are still widely used in current versions of the software.
- True Shape nesting + advanced options (grain direction, parts inside parts, non-rectangular sheets) significantly outperforms basic rectangular nesting.
- The combination of nesting + Knowledge Bases + Selection Rules creates a highly automated workflow from design to finished parts.
- The module works in both VisualCAD/CAM and RhinoCAM.
- Organize and clean geometry before creating nested layouts.
- Use true-shape nesting to maximize material utilization whenever possible.
- Balance grain direction constraints with nesting efficiency.
- Optimize cut sequencing to reduce unnecessary machine movement.
- Use nesting constraints and manual editing tools to fine-tune layouts.
- Reuse leftover material through remnant management workflows.

4.58 Machining Molds

Link: https://www.youtube.com/watch?v=jRJrfqqgvbc

This AMS webinar focuses on mold machining workflows using RhinoCAM and VisualCAD/CAM. The presentation explains how MecSoft CAM tools are used to machine mold cavities, cores, inserts, and complex free-form surfaces for injection molding and tooling applications. Topics include mold setup preparation, stock definition, roughing strategies, rest machining, finishing operations, pencil tracing, cutter selection, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving mold surface quality, reducing machining cycle time, minimizing manual polishing, and creating efficient CNC workflows for mold manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of mold machining challenges
4:00	Part Overview & Geometry Analysis	Mold core features, parting plane, draft angles, fillets
9:30	Recommended Tools & Stock Setup	Flat end mills vs ball mills, stock definition, work zero
14:30	Horizontal Roughing	Z-level roughing with step minimization and masking surfaces
26:00	Parallel Finishing (Primary Finishing Pass)	X & Y direction passes, step-over, tolerance settings
36:00	Horizontal Finishing (Sidewalls & Vertical Areas)	When and why to use it for mold walls
43:00	Parting Plane Handling & Guide Surfaces	Critical technique for maintaining parting line integrity
50:00	Pencil Trace / Valley Machining	Cleaning sharp corners and fillets
56:00	Simulation, Post-Processing & Best Practices	Full job simulation and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Never Machine Directly to the Parting Plane with a Flat End Mill** — Always leave 0.010"–0.025" stock on the parting plane during roughing and early finishing. Finish it later with a ball mill or corner radius mill to maintain tangent contact and protect the parting line.
2. **Use Guide Surfaces for Parting Line Integrity** — Add temporary guide surfaces (extruded or revolved) that extend past the parting plane. This forces the tool to cut **past** the parting line in X/Y while staying tangent in Z, preventing the tool from riding along or degrading the parting line edge.
3. **Run Parallel Finishing in Two Directions (0° + 90°)** — This dramatically improves surface finish on contoured mold surfaces and reduces the need for hand polishing.
4. **Step Minimization is Extremely Effective for Molds** — Combine roughing + re-roughing into a single operation by enabling step minimization (e.g., 25–50% of tool diameter). This saves significant time on deep cavities.
5. **Masking Surfaces Protect Critical Areas** — Create masking surfaces to prevent the tool from entering pockets, holes, or areas you want to leave stock on during roughing.
6. **Pencil Trace / Valley Machining for Sharp Corners** — Use this feature to clean up sharp internal corners and fillets that standard parallel or horizontal finishing cannot reach cleanly.
7. **Ball Mills for Finishing Only** — Use flat end mills for all roughing (faster calculation and material removal). Switch to ball mills only for final finishing passes.
8. **Clear Flats Option** — Enable “Clear Flats” in horizontal finishing operations to ensure the tool fully machines flat areas instead of lifting prematurely.
9. **Offset Curves for Safe Finishing Near Edges** — When finishing near parting lines or draft walls, offset your containment curves outward by tool radius + clearance to keep the tool from gouging or rolling off the edge.
10. **Simulation is Non-Negotiable** — Always run full cut material simulation with fine accuracy before posting, especially when using guide surfaces or multiple finishing directions.

Key Takeaways

This webinar is an excellent **practical guide** for anyone programming injection molds. The biggest emphasis is on **protecting the parting line** — one of the most critical (and mistake-prone) areas in mold machining.

Core workflow highlighted:

- The techniques shown in this webinar remain highly relevant and are still used in current versions of RhinoCAM and VisualCAD/CAM.
- Rough with flat end mills + step minimization.

- Parallel finish in two directions.
- Use guide surfaces + proper stock on parting plane.
- Pencil trace / horizontal finishing for final cleanup.
- Prepare mold geometry and machining regions carefully before programming.
- Use aggressive roughing strategies before fine finishing operations.
- Apply re-roughing and rest machining to remove leftover cavity material efficiently.
- Optimize stepover values for improved mold surface finish quality.
- Match cutter geometry carefully to cavity detail size and surface requirements.
- Validate mold machining operations with simulation before posting NC code.

4.59 Mesh Machining

Link: https://www.youtube.com/watch?v=bVUh_fnoQl4

This AMS webinar focuses on machining mesh geometry in RhinoCAM and VisualCAD/CAM. The presentation explains how polygon mesh models such as STL and scanned geometry can be prepared, optimized, and machined effectively using MecSoft CAM tools. Topics include mesh import workflows, mesh cleanup, triangle density management, stock setup, roughing operations, finishing strategies, stepover optimization, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining quality, reducing calculation time, minimizing mesh-related issues, and creating efficient CNC machining workflows when working with tessellated and scanned geometry.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of mesh machining challenges and solutions
3:00	Guest Presentation: Jim Lohmann	Real-world experience machining large ZBrush meshes for architectural woodcarving and historic restoration
32:00	Curve Extraction from Meshes	Using Mesh Outline and Flat Area Regions to create machining boundaries
38:00	2.5-Axis & 3-Axis Machining on Meshes	Pocketing, facing, roughing, and parallel finishing on mesh geometry
45:00	4-Axis Machining on Meshes	Programming rotary axis parts using re-meshed geometry
50:00	5-Axis Considerations with Meshes	Using meshes as gouge-check geometry while toolpaths are driven by NURBS surfaces
55:00	Applications & Q&A	Orthotics, architectural work, mold making, and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Always Repair Meshes Before Machining** — Use the Auto Fix tool to automatically close holes, correct normals, and fix self-intersections. Machining an un repaired mesh often leads to poor toolpaths or crashes.
2. **Decimate (Reduce) Large Meshes** — Meshes with millions of triangles (common from ZBrush or 3D scans) dramatically slow down toolpath calculation. Use the Mesh Reduction tool to lower triangle count while preserving important detail. MecSoft's reduction tool generally performs better than Rhino's native tools for machining purposes.
3. **Re mesh for Better Surface Finish** — After reducing a mesh, use the Re mesh command to create more uniform triangles. This often improves the quality of parallel and 3D offset finishing toolpaths.
4. **Extract Curves from Meshes for Boundaries** — Since meshes don't have clean edges like NURBS, use Mesh Outline and Flat Area Regions tools to generate 2D curves you can use for containment, pocketing, or profiling boundaries.
5. **Meshes Work Well with Standard 3-Axis Strategies** — Z-level (Horizontal) Roughing + Parallel or 3D Offset Finishing is a reliable workflow for most mesh models.

6. **Use Meshes as Gouge Check in 5-Axis** — For true 5-axis work, you can drive toolpaths from an underlying NURBS surface while using the mesh as collision/gouge-check geometry. This is useful when you have a high-quality surface model but also need to respect scanned or artistic mesh data.
7. **Verify Units Immediately After Import** — Many mesh files come in the wrong units (mm vs inches). Always check and correct units right after importing to avoid dimensional errors.
8. **Boolean Operations on Meshes** — The Mesh Module supports Boolean operations (union, subtract, intersect) on meshes, which is useful for creating mold cavities or modifying imported scan data.
9. **Jim Lohmann's Workflow Tip** — For large architectural carvings, he typically:
    1. Imports the high-poly ZBrush mesh
    2. Reduces it significantly using the Mesh Module
    3. Machines roughing passes aggressively
    4. Finishes with smaller tools and minimal hand carving

Key Takeaways

This webinar is one of the best resources for understanding how to work with mesh geometry in MecSoft CAM. It clearly shows that meshes are no longer second-class citizens — with the Mesh Module, you can reliably machine scanned data, ZBrush models, and artistic reliefs.

Biggest lessons:

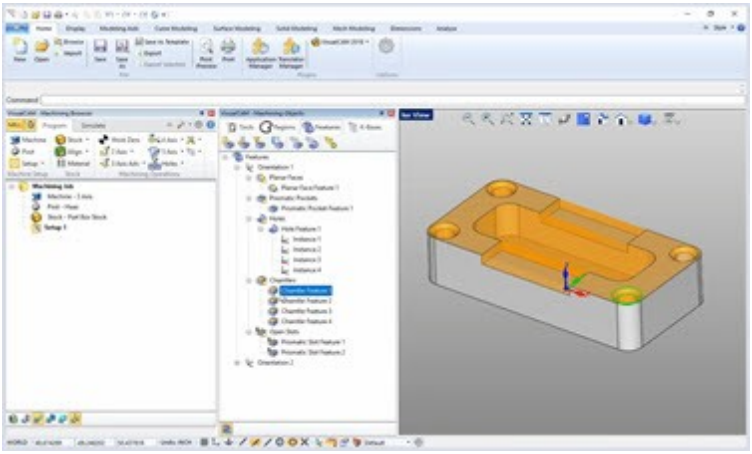
- Repair + Decimate + Re mesh before machining.
- Standard 2.5/3-axis strategies work very well on optimized meshes.
- The Mesh Module dramatically improves performance and usability with large or imperfect mesh files.
- Repair and clean mesh geometry before generating toolpaths.
- Reduce excessive mesh density to improve CAM performance.
- Optimize roughing workflows before fine finishing operations.
- Balance tolerance settings against machining cycle time.
- Verify mesh machining workflows using simulation before production.
- Organize mesh setups carefully to simplify machining workflows.

4.60 Feature Machining

Link: https://www.youtube.com/watch?v=5MvOqdybBlw

This AMS webinar focuses on feature detection and automated machining workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how machinable features such as holes, pockets, slots, bosses, and planar regions can be identified automatically from CAD models and used to streamline CAM programming. Topics include automatic feature recognition, machining templates, operation automation, tool selection, hole machining, setup organization, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for reducing repetitive programming work, improving workflow consistency, minimizing setup errors, and accelerating CNC machining workflows for production environments.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of Feature Detection & Machining
3:00	Automatic Feature Detection (AFD)	How the software analyzes solid models to find features
10:00	Automatic Feature Machining (AFM)	Mapping detected features to operations using Knowledge Bases
16:00	Feature Types Supported	Holes, pockets (closed/open), bosses, slots, chamfers, T-slots
22:00	Interactive Feature Detection & Machining	Manually selecting features and generating toolpaths on demand
34:00	Customizing the AFM Knowledge Base	Adding operations, reordering, and saving custom KBs

Time	Topic / Section	Description
40:00	Drilling Automation	Automatic hole grouping, tool selection, and cycle generation
48:00	Q&A and Limitations	Questions about 3D features, base plates, and future improvements

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Feature-Based Machining is Extremely Powerful for Prismatic Parts** — It can automatically detect and machine holes, pockets, slots, and chamfers with minimal manual intervention. This is especially useful for parts with many similar features.
2. **Customize the Knowledge Base for Your Workflow** — The default AFM Knowledge Base is limited. You should create your own custom Knowledge Base that includes the exact operations you want (e.g., add drilling cycles for holes, specific roughing/finishing strategies for pockets, etc.).
3. **Add Hole Features to Milling Knowledge Bases** — By default, many milling Knowledge Bases do not include drilling operations. You must manually add hole feature types and assign the appropriate drilling operations (Spot Drill → Drill → Profile, etc.).
4. **Interactive Feature Machining is Great for Complex Parts** — When automatic detection misses features or you want more control, use **Interactive Feature Detection**. Simply select a face or feature and the software suggests appropriate operations.
5. **Reorder Operations in the Knowledge Base** — The order of operations in your AFM Knowledge Base matters. Put roughing operations before finishing operations, and group similar features together for cleaner output.
6. **Feature Detection Works Best on Clean Solid Models** — The software detects features most reliably on clean, well-modeled solid geometry. Avoid using meshes or poorly defined models for automatic feature detection.
7. **Use Separate Knowledge Bases for Different Part Types** — Create different AFM Knowledge Bases for different classes of parts (e.g., one for aluminum plates, one for steel molds, one for wood components) so you can quickly switch workflows.
8. **Automatic Tool Selection from Library** — When using AFM with holes, the software can automatically select the correct drill size from your tool library based on hole diameter.

Key Takeaways

This webinar introduced one of the **highest-productivity features** in MecSoft CAM: **Feature Detection & Machining**. When combined with well-customized Knowledge Bases, it can reduce programming time dramatically on prismatic parts with repetitive features.

Biggest value:

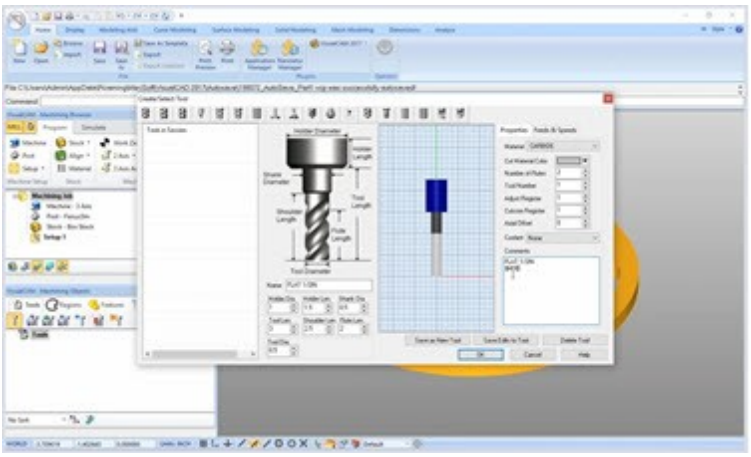
- The concepts shown in this webinar remain foundational and are still widely used today.
- Automatic detection of holes, pockets, slots, and chamfers.
- One-click toolpath generation using Knowledge Bases.
- Highly customizable through the AFM Knowledge Base editor.
- Organize CAD geometry carefully before running feature detection.
- Verify automatically detected features before generating toolpaths.
- Standardize tooling and machining templates for reliable automation.
- Use feature detection workflows for repetitive production parts.
- Regenerate operations after CAD geometry changes.
- Validate automatically generated operations with simulation before posting NC code.

4.61 Cutting Tools

Link: https://www.youtube.com/watch?v=dqrU3sW27BE

This AMS webinar focuses on cutting tools and tooling strategies used in RhinoCAM and VisualCAD/CAM machining workflows. The presentation explains how cutter geometry, tool selection, feeds and speeds, tool libraries, and machining parameters affect machining quality, efficiency, and tool life. Topics include end mills, ball mills, drills, specialty cutters, holder considerations, feeds and speeds, stepover optimization, simulation, and tooling best practices. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining performance, extending tool life, reducing cycle times, and selecting the proper cutting tools for real-world CNC applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of cutting tool management in MecSoft CAM
4:30	Creating Standard Tools	Ball mills, flat mills, corner radius, engraving, chamfer, taper, face mills, dovetail, lollipop, drills, taps, etc.
10:00	Defining Tool Properties	Geometry (diameter, flute length, shank, holder), material, color, registers (tool number, G43, D-code), axial offset, coolant, comments
18:00	Feeds & Speeds Setup	Manual entry vs loading from files, plunge/engage/retract rates
21:00	Tool Libraries – Saving & Loading	Default libraries, CSV export, library persistence, drag-and-drop selective loading
35:00	User-Defined (Custom Form) Tools	Creating custom tools from half-profile curves in CAD (tip at origin)
41:30	FAQ: Printing Tool Lists & G-Code Output	Printing from Tools tab + environment variable Vcam_post_write_tool_list = 1
47:00	FAQ: Feed Rate Limits & Tapping	Post-processor high/low limits and tapping feed rate calculation (Pitch × RPM)

Time	Topic / Section	Description
51:50	FAQ: Optimizing Machining Time	Adjusting step-over, depth of cut, and non-cutting feed rates
57:00	FAQ: Tool Change Points & Comments	Defining safe tool change positions + using $ prefix for G-code comments
1:02:00	FAQ: Cutter Compensation (G41/G42/G40)	Requirements (linear entry), post settings, and how it works with CAM compensation
1:07:00	Conclusion & Resources	Cutting Tools Workbook (100-page guide) for AMS subscribers

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Use the $ Prefix for Executable Comments** — Any comment starting with $ in a tool or operation is output as actual G-code (e.g., $M00 for optional stop) instead of a comment line.
2. **Tool Registers Are Automatically Linked** — The software automatically sets the Height Offset (G43) and Cutter Compensation (D-code) registers to match the tool number. You can override this if needed.
3. **Library Persistence Can Be Controlled** — The last loaded tool library is remembered by default. You can disable this behavior in Cutting Tool Preferences if you prefer starting fresh each time.
4. **Drag-and-Drop Tools from Libraries** — You don't have to load an entire library. Simply drag individual tools from a saved library into your current project.
5. **User-Defined Tools Must Have the Tip at the Origin** — When creating custom form tools, the tip of the half-profile curve must be placed at (0,0,0) in your CAD system for correct toolpath calculation.
6. **Environment Variable for Tool List in G-Code** — Set Vcam_post_write_tool_list = 1 in your system environment variables to automatically output a tool list at the beginning of every posted NC file.
7. **Post-Processor Feed Limits** — If your posted feed rates seem capped, check the high and low feed rate limits defined in your post-processor. The post will never exceed these values.

8. **Tapping Feed Rate Formula** — Most controllers expect tapping feed rate = Pitch × RPM. MecSoft posts use macros like cycle_ipr or cycle_tpi to output the correct format.
9. **Tool Change Points for Safety** — Define a safe Tool Change Point (X, Y, Z) in the Machine Setup dialog. The post can then automatically move the tool to this position before every tool change.
10. **Cutter Compensation Requires Linear Entry** — G41/G42 only works reliably when the tool approaches on a straight line (tangent to the first cut). Arcs or ramps during engagement often cause the controller to ignore compensation.
11. **Print Tool Lists Directly** — In the Tools tab, you can generate a printable list of all loaded tools (or save it as PDF) for shop floor use or setup sheets.

Key Takeaways

This webinar is one of the best resources for mastering cutting tool management in MecSoft CAM. It goes far beyond basic tool creation and answers the real-world questions that users ask support most often.

Biggest value:

- The 100-page Cutting Tools Workbook mentioned at the end of the webinar is still available to all active AMS subscribers and is highly recommended as a companion resource.
- Proper tool property setup (registers, coolant, comments).
- Efficient library management and user-defined tools.
- Understanding how the post-processor interacts with tool data (feeds, cutter compensation, tool change points).
- Match cutter geometry carefully to each machining operation.
- Standardize tooling libraries for consistent and efficient programming workflows.
- Use larger cutters whenever possible for aggressive roughing operations.
- Optimize stepover and stepdown values to balance finish quality and machining time.
- Verify holder clearance and tool reach before finalizing setups.
- Monitor tool wear regularly to maintain machining accuracy and surface finish quality.
- Use simulation tools to validate cutter motion and setup safety before machining.

4.62 3 Axis Best Practices

Link: https://www.youtube.com/watch?v=IuIWk1pi6Sk

This AMS webinar focuses on best practices for 3 Axis machining in RhinoCAM and VisualCAD/CAM. The presentation explains practical machining workflows used to improve surface finish quality, reduce machining time, optimize cutter performance, and streamline CNC programming for complex 3D parts. Topics include geometry preparation, stock setup, roughing

strategies, re-roughing, finishing operations, cutter selection, tolerance settings, simulation, and post processing. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining reliability, minimizing air cutting, reducing manual finishing work, and creating efficient 3 Axis machining workflows for real-world manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview
2:10	CAD Geometry Types & File Formats	Solids vs Surfaces vs Meshes; Native vs STEP vs IGES; translation issues
7:20	Demo: Importing Different File Formats	IGES vs STEP vs Mesh comparison and common problems
13:55	Best Export Settings	Using Trim Surface (Type 144) for IGES and STEP recommendations
16:30	Machining Tolerances	Global tolerance, Arc Fitting (2× rule), Part Faceting, Inlet/Outlet tolerances
25:20	Demo: Adjusting Tolerances & Stock to Leave	Practical settings for roughing and finishing
36:30	Stock to Leave Strategies	Positive vs Negative stock values and when to use each

Time	Topic / Section	Description
44:50	Surface Finish Best Practices	Factors affecting finish and use of containment
49:10	3-Axis Operation Overview	Horizontal Roughing, Parallel Finishing, Horizontal Finishing, Spiral & Radial
53:55	Demo: Parallel Finishing	Angle of cut (0° vs 90°), Z-containment, step-over
1:01:45	Demo: Horizontal Finishing	Steep walls, “Optimized XY Machining”, Clear Flats
1:11:15	Demo: Spiral & Radial Machining	Circular features, center point selection, minimum radius
1:14:10	Conclusion & Q&A	Final recommendations and resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Solids > Surfaces > Meshes** — Always prefer **solid models** for 3-axis machining. They provide the best accuracy, feature detection, and watertight boundaries. Use surfaces only when necessary. Avoid meshes unless you have no other choice (scans, artistic work).
2. **STEP is Preferred Over IGES** — STEP files preserve topology (connected edges) and create watertight poly-surfaces. IGES files frequently produce floating, disconnected surfaces that cause tool diving, double marks, or poor finishes.
3. **Use Trim Surface (Type 144) When Exporting IGES** — This export option in most CAD systems significantly improves IGES translation quality into MecSoft CAM.
4. **Arc Fitting Tolerance = 2× Global Tolerance** — This is the golden rule for best results. It converts linear moves into smooth G2/G3 arcs while keeping file size manageable.
5. **Negative Stock to Leave is Powerful** — Use negative stock (e.g., -0.010") when you need to **over-cut** (shrink) a surface — excellent for parts that will receive coatings, plating, or need a tight interference fit. Never exceed the smallest radius of your finishing tool.
6. **Run Parallel Finishing in Two Directions** — Performing one pass at 0° and a second at 90° dramatically reduces visible scalloping and improves surface finish on most parts.
7. **“Optimized XY Machining” in Horizontal Finishing** — This option (newer at the time) allows Horizontal Finishing to also machine near-flat areas between Z-levels, giving you the best of both Horizontal and Parallel strategies in one operation.

8. **Use Containment Regions Generously** — Don't let the tool machine the entire part if you only need a specific area. Containment dramatically reduces calculation time and prevents unwanted toolpaths.
9. **Spiral Machining for Circular Areas** — When you have circular or near-circular regions, Spiral machining creates a continuous path with no retracts — often the cleanest and fastest option.
10. **Clear Flats Option** — Enable this in Horizontal Roughing or Finishing when you want the tool to fully machine flat horizontal surfaces instead of leaving small cusps.

Key Takeaways

This webinar remains one of the best **foundational resources** for 3-axis best practices in MecSoft CAM. The core message is:

Better input geometry + smart tolerances + proper operation selection = dramatically better results with less handwork.

The biggest productivity and quality gains come from:

- Using solid models and STEP files whenever possible
- Following the 2× Arc Fitting rule
- Strategically using containment and running finishing passes in two directions
- Choosing the right operation for the geometry (Horizontal for steep walls, Parallel for shallow areas, Spiral for circular regions).
- Use clean and organized CAD geometry before programming toolpaths.
- Optimize roughing workflows before refining finishing operations.
- Apply re-roughing and rest machining to remove leftover material efficiently.
- Optimize stepover values for improved surface finish quality.
- Match machining tolerance settings carefully to part requirements.
- Verify tool reach and holder clearance before machining.
- Group operations logically by tool and machining strategy to improve workflow efficiency.
- Validate machining operations using simulation before posting NC code.

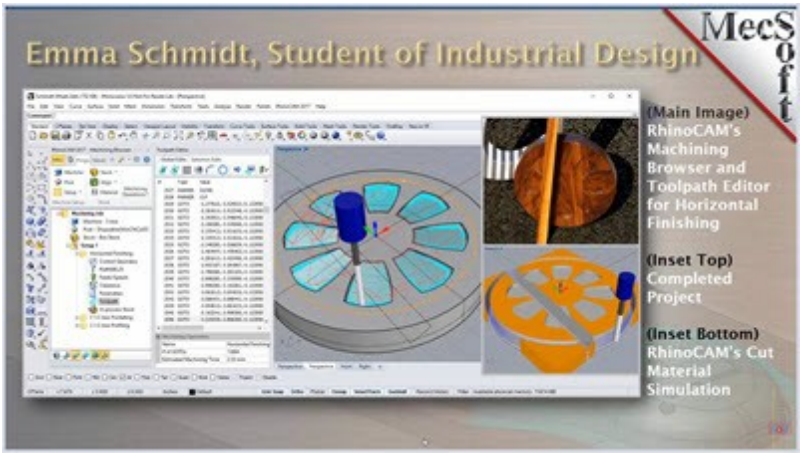
4.63 CAM in Education

Link: https://www.youtube.com/watch?v=CKzD1puZVr0

This AMS webinar focuses on the use of RhinoCAM and VisualCAD/CAM in educational environments for teaching CAD/CAM, CNC machining, engineering, and manufacturing technology. The presentation explores how schools, maker spaces, technical programs, and universities use MecSoft CAM software to introduce students to real-world manufacturing

workflows. Topics include classroom setup, CAD/CAM curriculum development, beginner machining projects, simulation, CNC machine safety, educational licensing, and student project workflows. Throughout the webinar, the presenters also share practical teaching tips for helping students build hands-on manufacturing skills while learning CAD design and CNC programming concepts.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Speakers	Overview and introduction of Bill Bancroft & Emma Schmidt
3:00	Bill Bancroft's Background	Transition to architectural mill-work and long-term use of RhinoCAM since 2008
7:50	Emma Schmidt's Student Project	Memphis-inspired console table design, challenges, and solutions
15:10	Live RhinoCAM Demo	Machining the decorative "wheel" component (Horizontal Finishing, Access Pocketing, Profiling)
26:00	Educational Licensing Options	Lab Licenses vs Campus CAM, pricing, and AMS benefits
34:30	Q&A Session	Questions about other projects, boring bits, and software compatibility

Time	Topic / Section	Description
46:00	Conclusion & Resources	Final thoughts and available educational resources

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Bridges & Tabs Keep Stock Secure** — Use the built-in Bridges/Tabs feature in 2.5-axis profiling to automatically hold nested parts to the sheet. This eliminates the need for manual tabs and prevents parts from moving during cutting.
2. **Adjust Tolerances Between Prototype and Final Material** — When moving from a prototype (e.g., plywood) to final material (e.g., walnut), tighten or loosen tolerances by a few thousandths of an inch to achieve a perfect fit, especially on complex joints like graduated finger joints.
3. **Use 3-Axis Horizontal Finishing for Angled Surfaces** — For parts with angled or non-planar features (like the serpentine stretcher), 3-axis Horizontal Finishing provides better control than standard parallel finishing.
4. **Laminated Plywood for Short-Grain Problems** — When solid wood has short grain issues (common in curved or serpentine parts), switch to multi-layer Baltic birch plywood. It machines cleanly and provides better structural integrity.
5. **Feature-Based Workflows for Repetitive Elements** — The "wheel" component was efficiently divided using Access Pocketing with a rectangular containment boundary — a smart way to handle complex decorative elements.
6. **Educational Licensing is Very Flexible** — Schools can choose:
    - Lab Licenses (network-restricted to one classroom)
    - Campus CAM (unlimited network-wide access, ideal for laptops and multiple locations)
7. **AMS Provides Excellent Educational Support** — The Annual Maintenance Subscription includes free version upgrades and access to the full CAM Jam video library (100+ videos, 10+ hours of content) — extremely valuable for both students and instructors.
8. **Public Domain Projects Available** — MecSoft offers free public domain projects (including an F1 car project designed for high schools) to help educators get started quickly.
9. **Real Student Projects Increase Engagement** — Giving students the opportunity to design and machine functional, professional-quality furniture (like Emma's console table) dramatically improves learning outcomes and portfolio quality.

Key Takeaways

This webinar is an excellent real-world example of how RhinoCAM can be successfully integrated into higher education. The combination of a strong industry partner (Bill Bancroft), motivated students (Emma Schmidt), and practical toolpath strategies (Horizontal Finishing + Access Pocketing + Bridges/Tabs) demonstrates that professional CAM software is very accessible for design and manufacturing students.

Biggest lessons:

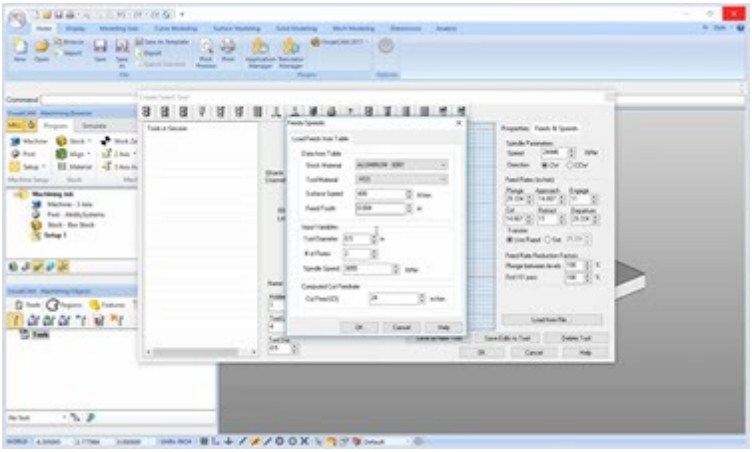
- The session proves that students can produce professional-grade work when given the right tools and guidance.
- Real, meaningful projects (furniture, architectural elements) create high student engagement.
- Features like Bridges/Tabs and Horizontal Finishing make complex parts much more manageable for students.
- Flexible educational licensing and strong support resources (CAM Jam, public projects) lower the barrier for schools.
- Start students with simple and achievable machining projects.
- Use standardized classroom workflows and reusable project templates.
- Teach simulation workflows before students operate CNC machines.
- Connect CAD design directly to real manufacturing projects.
- Use reusable machining templates to simplify instruction.
- Focus on hands-on CNC machining experience to reinforce learning.
- Organize classroom workflows to improve machine access and project efficiency.
- Prepare students for real-world manufacturing and engineering careers through practical CAD/CAM training.

4.64 Feeds and Speeds

Link: https://www.youtube.com/watch?v=2-S_YoRchJw

This AMS webinar focuses on understanding and applying feeds and speeds in RhinoCAM and VisualCAD/CAM machining workflows. The presentation explains how spindle speed, feed rate, chip load, cutter engagement, material type, and machining strategy directly affect machining performance, surface finish, tool life, and CNC efficiency. Topics include feeds and speeds fundamentals, material considerations, cutter selection, chip load calculations, roughing and finishing adjustments, built-in feeds and speeds calculators, simulation, and troubleshooting. Throughout the webinar, the presenters also share practical CAM programming tips for improving machining efficiency, extending tool life, reducing cycle times, and optimizing cutting performance for real-world CNC machining applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Why feeds and speeds matter and what will be covered
3:45	Feeds & Speeds Fundamentals	Chip load, spindle speed (RPM), feed rate (IPM), and material/tool relationships
9:20	Tool Library Setup	Diameter, number of flutes, tool material, and chip load values
16:40	How MecSoft Calculates RPM & Feed Rate	Formulas and automatic calculation process
23:10	Plunge, Engage, Retract & Cut Rates	Recommended percentages and when to adjust them
31:50	Common Mistakes & Problems	Overly aggressive settings, wrong chip load, ignoring tool material
40:00	Live Demo: Setting Up Tools & Generating Toolpaths	Practical example with aluminum and wood
50:30	Post-Processor Output & Verification	How feeds and speeds appear in G-code

Time	Topic / Section	Description
57:00	Q&A and Best Practices Wrap-Up	Final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Chip Load is the Foundation** — Always start with the correct chip load for your tool diameter and material. MecSoft uses this value to calculate both RPM and feed rate. Using the wrong chip load is the #1 cause of poor tool life or broken tools.
2. **Start Conservative, Then Increase** — Begin with 60–70% of the recommended chip load and gradually increase while monitoring tool wear and surface finish. This is especially important when working with new materials or machines.
3. **Plunge Rate Should Be Much Lower** — Set plunge/engage rates to **25–40%** of the normal feed rate. Many users leave this at 100%, which causes tool breakage on entry.
4. **Use Different Rates for Different Materials** — Create separate tool libraries or Knowledge Bases for aluminum, steel, wood, and plastics. The optimal chip load and RPM vary dramatically between materials.
5. **Tool Material Matters** — Carbide tools can run significantly faster (higher RPM and feed) than HSS tools. Always select the correct tool material in the library.
6. **Verify in the Post** — After posting G-code, always check that the actual feed rates and spindle speeds in the NC file match what you expect. Some post-processors apply limits that can override your settings.
7. **Use the Built-in Calculator** — MecSoft's tool library includes a simple calculator that shows the resulting RPM and IPM based on your chip load input. Use it as a quick sanity check.
8. **Engage/Retract Rates** — Set engage and retract rates slightly lower than the cut rate (typically 70–80%) to reduce tool stress during direction changes.
9. **Document Your Settings** — Save successful feeds and speeds combinations as **Knowledge Bases** so you can reuse them on future jobs with the same material and tool.
10. **Monitor and Adjust** — The best feeds and speeds are often found through testing. Start safe, run a test cut, and adjust based on sound, chip formation, and tool wear.

Key Takeaways

This webinar remains one of the best resources for understanding how **feeds and speeds** actually work inside MecSoft CAM. The core message is:

Biggest lessons:

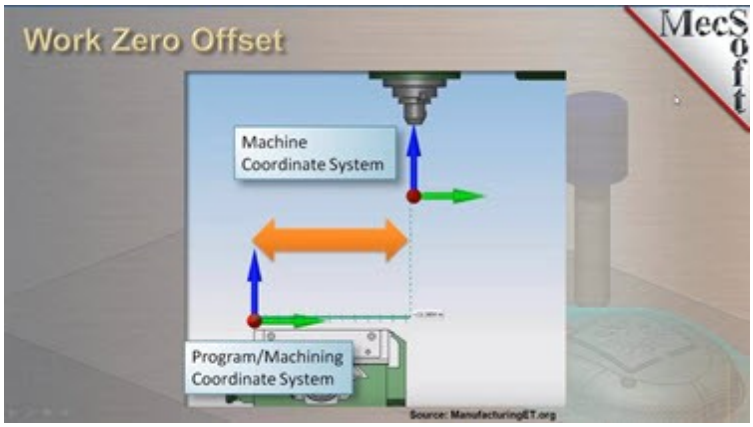
- The techniques and recommendations shown in this webinar are still fully applicable in current versions of RhinoCAM and VisualCAD/CAM.
- Good feeds and speeds come from proper tool setup + realistic chip load values + conservative entry rates.
- Chip load is the single most important input.
- Plunge/engage rates are frequently set too high.
- Creating material-specific tool libraries or Knowledge Bases saves enormous time and reduces errors.
- Start with conservative machining parameters and optimize gradually.
- Match feeds and speeds carefully to cutter material and stock material.
- Use chip load calculations to maintain proper cutter engagement.
- Use the built-in Feeds and Speeds Calculator as a starting point, not a final answer.
- Reduce feed rates slightly during finishing operations for improved surface quality.
- Customize material libraries to match your machine and tooling environment.
- Monitor spindle load and chip evacuation during initial test cuts.

4.65 Coordinate Systems

Link: https://www.youtube.com/watch?v=x184fnSWqM0

This AMS webinar focuses on coordinate systems and machining setup workflows in RhinoCAM and VisualCAD/CAM. The presentation explains how coordinate systems affect machining orientation, work offsets, toolpath alignment, and multi-setup CNC workflows. Topics include machine coordinate systems, work coordinate systems (WCS), setup alignment, stock orientation, indexed machining, post processing, and simulation. Throughout the webinar, the presenters also share practical CAM programming tips for reducing setup errors, improving machining accuracy, simplifying multi-sided machining workflows, and organizing CNC programs efficiently for real-world manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Why coordinate systems matter in CAM
3:45	Coordinate System Definitions	World Coordinate System (WCS), Machine Coordinate System (MCS), Program/Machining Coordinate System, and Work Zero Offset (G54/G55)
9:20	VisualCAD/CAM Demonstration	Importing parts, using "Orient Part", setting World Coordinates, and defining Work Zero
18:40	RhinoCAM Demonstration	Handling parts far from origin, Orient Part tool, and transforming both part + stock
38:00	Turning Coordinate Mapping	How World X/Y map to Turn Z/X and proper part orientation for lathe work
44:30	4-Axis Indexed Machining Demo	Creating Rotate Table Setup and outputting rotary axis commands (A180 example)
51:00	Q&A and Best Practices Wrap-Up	Common questions and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Always Align the Part's Z-Axis to the Machining Face** — Before programming, make sure the machining face is normal to the World Z-axis. Use the "Orient Part" command in the CAM browser or CAD transformation tools (Orient C-Plane using three points) to rotate the part correctly.
2. **Two Main Approaches to Setting the Origin**:
    - Move the geometry so the desired machining origin coincides with the World Origin (0,0,0).
    - Keep the geometry in place and use a Work Zero to define an offset origin. This is often cleaner and preserves design intent.
3. **For Turning: Back Face Must Be at Origin** — The back face of the part must be placed at (0,0,0) with the part in the first quadrant. World X becomes Turn Z (spindle axis) and World Y becomes Turn X.

4. **Use Multiple Setups for Multi-Sided Parts** — Create a new Setup for each orientation (top, bottom, side, etc.). Each Setup can have its own coordinate system and Work Zero. This keeps everything in one file and makes simulation much more reliable.
5. **Work Zero Offset = G54/G55** — The Work Zero you define in CAM becomes the G54 (or G55, G56, etc.) offset on the machine. Always verify this matches your machine's setup.
6. **CAM Transformations Can Move Both Part and Stock** — Unlike pure CAD transforms, RhinoCAM's "Orient Part" and similar tools can simultaneously transform the part and the stock geometry — very useful when the part was modeled far from the origin.

Key Takeaways

This webinar is one of the most important foundational sessions in the MecSoft AMS library. Many support issues (parts machining in the wrong place, incorrect depths, or misaligned features) stem from misunderstanding coordinate systems.

Biggest lessons:

- The concepts covered in this webinar remain fully relevant and are still the foundation of correct setup in all current versions of RhinoCAM and VisualCAD/CAM.
- Always ensure the machining face is normal to Z before programming.
- Choose between moving geometry or using Work Zero based on your workflow.
- Multiple Setups are the cleanest way to handle multi-sided parts.
- Turning has very specific orientation rules that must be followed.
- Define the primary work coordinate system before creating machining operations.
- Use clear naming conventions for setups and coordinate systems.
- Align work coordinates to fixtures whenever possible for repeatability.
- Verify G54-G59 work offsets carefully before posting NC code.
- Group machining operations logically by coordinate system and setup.
- Simulate all setup orientations before machining production parts.
- Organize operation trees carefully for multi-sided and indexed machining workflows.

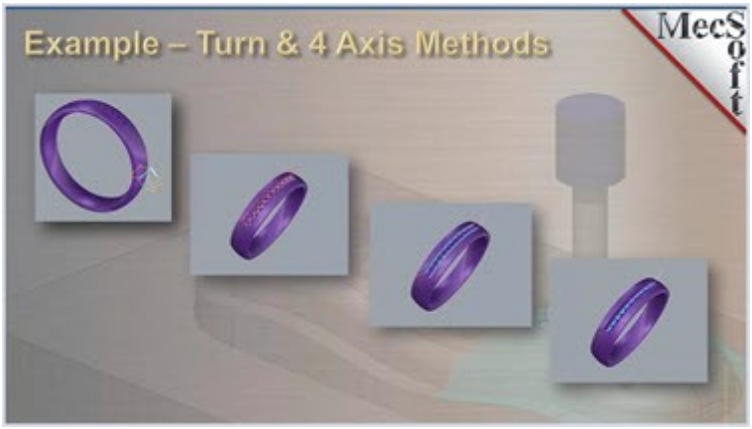
4.66 4 Axis Jewelry

Link: https://www.youtube.com/watch?v=5dpQe1jR8eA

This AMS webinar focuses on advanced jewelry machining workflows using RhinoCAM and VisualCAD/CAM. Building on foundational jewelry machining concepts, the presentation explores practical CAM strategies for machining rings, pendants, relief models, and detailed ornamental parts using 2½ Axis, 3 Axis, and rotary machining workflows. Topics include jewelry model preparation, fine-detail tooling, roughing and finishing strategies, rotary machining, stepover optimization, surface finish control, simulation, and post processing. Throughout the webinar, the

presenters also share practical CAM programming tips for improving detail quality, minimizing polishing work, reducing machining time, and creating efficient CNC workflows for jewelry manufacturing applications.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction and Housekeeping	Overview of the session
1:09	Recap of Part 1	2-axis, 2.5-axis, and basic 4-axis methods for pendants and rings
11:37	4-Axis Wrap Machining for Bands	Unwrapping cylindrical geometry for pocketing and profiling
31:00	Programming Ring Blanks using Turn Module	OD/ID roughing and finishing on ring blanks
42:00	Milling Module for Ring Details	4-axis drilling and engraving on turned blanks
50:00	True 4-Axis Machining using Drive Surface Method	Continuous 4-axis for complex undercuts in mold cavities

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **4-Axis Wrap Machining is Ideal for Constant-Radius Parts** — For ring bands and similar cylindrical jewelry, use **4-Axis Wrap** (Pocketing or Profiling). The software automatically

unwraps the geometry, computes the toolpath on a flat plane, and wraps it back — dramatically simplifying programming.

2. **Always Define the Correct 4-Axis Machine Configuration** — Set up your machine as a **rotary table** (A-axis) with the rotation center properly defined. This ensures the post-processor outputs the correct rotation angles.
3. **Use "Stock from Selection" for Accurate Ring Blanks** — When the ring blank has an irregular outer shape, use **Stock from Selection** (Professional/Premium) instead of a simple cylinder. This gives you much more accurate in-process stock tracking.
4. **Turn Module is Extremely Efficient for Ring Blanks** — Use the **TURN module** to machine the OD and ID of ring blanks with proper insert orientation (OD Forward/Backward and ID tools). The software automatically handles relief angle collision checking.
5. **Switch Between Turn and Mill Modules Seamlessly** — After turning the basic ring blank, switch back to the **MILL module** to add stone-setting holes, engraving, or decorative details using 4-axis drilling and engraving operations.
6. **Create User-Defined Tools for Custom Jewelry Profiles** — For unique stone settings or decorative cavities, draw the exact tool profile in your CAD system and load it as a **User-Defined Tool**. This gives you precise control that standard tools cannot achieve.
7. **Drive Surface Method for True 4-Axis Undercuts** — When features extend below the rotation center (common in mold cavities), use **Drive Surface Machining** with the **"Project to Geometry"** option enabled. This allows the tool to stay normal to a drive surface while projecting the path onto complex geometry.
8. **"Across Axis" vs "Along Axis" Cut Patterns** — In true 4-axis operations, try both patterns. "Across Axis" often produces better finish on curved jewelry surfaces, while "Along Axis" can be faster for certain geometries.
9. **Inverse Time Feed (G93) for Smooth 4-Axis Motion** — For controllers that support it, enable inverse time feed output in the post-processor. This produces much smoother coordinated rotary + linear motion on complex 4-axis jewelry toolpaths.
10. **Always Simulate Full 4-Axis Motion** — Because jewelry parts are small and delicate, always run full machine simulation (including rotary axis movement) before posting to catch any collisions with the tool holder or fixture.

Key Takeaways

This webinar is an excellent **practical guide** for jewelry manufacturers using MecSoft CAM. It clearly demonstrates how to combine multiple modules (MILL + TURN + advanced 4-axis) to efficiently produce high-quality jewelry components and molds.

Biggest lessons:

- 4-Axis Wrap + Drive Surface methods open up powerful new capabilities for rings and molds.
- The Turn module is highly effective for ring blanks and should be used before adding details in the Mill module.
- Proper machine configuration and simulation are critical when working with small, high-value jewelry parts.
- Simplify CAD geometry before generating detailed jewelry toolpaths.
- Match cutter diameter carefully to the smallest required feature size.
- Optimize stepover values to balance finish quality and machining time.
- Use smaller step-downs when machining delicate or fragile geometry.
- Reduce feed rates during finishing operations to improve surface quality.
- Simulate every machining setup before cutting expensive materials.
- Use rotary workflows for rings and cylindrical jewelry components.

4.67 CAM in Education

Link: https://www.youtube.com/watch?v=I98NgfuZr-Y

This AMS webinar focuses on how RhinoCAM and VisualCAD/CAM are used in educational environments to teach CAD/CAM, CNC machining, engineering, and manufacturing workflows. The presentation highlights classroom implementation strategies, student machining projects, CNC training workflows, and educational applications at schools and universities including South Dakota State University (SDSU). Topics include CAD modeling, CAM programming, simulation, CNC safety, classroom workflows, educational licensing, and real-world student manufacturing projects. Throughout the webinar, the presenters also share practical teaching tips for helping students build manufacturing skills while preparing for careers in machining, engineering, product design, and advanced manufacturing industries.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview of CAM in education
2:45	Why MecSoft CAM for Schools	Ease of use, affordability, strong support, and industry relevance
7:30	Guest Educator Presentation	Real classroom experience and curriculum integration
14:00	Student Project Showcase	Examples of student work (furniture, jewelry, molds, or signage)
20:00	Live Toolpath Demonstration	RhinoCAM or VisualCAD/CAM in action on student geometry
28:00	Educational Licensing Options	Lab Licenses, Campus CAM, instructor keys, and pricing
35:00	AMS Benefits for Educators	CAMJam video library, monthly webinars, technical support
40:00	Q&A Session	Common questions from teachers and administrators
46:00	Conclusion & Resources	Final thoughts and links to educational materials

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Start with Simpler Projects** — Many educators begin with 2.5-axis projects (signs, plaques, simple molds) before moving to full 3-axis work. This builds student confidence and reduces frustration.
2. **Knowledge Bases Save Teaching Time** — Teachers can create and save Knowledge Bases with pre-configured tools, tolerances, and strategies. Students can then load the correct KB and focus on geometry selection rather than parameter setup.
3. **Bridges/Tabs Are Essential for Student Projects** — Always use the Bridges/Tabs feature when cutting multiple parts from a single sheet. This prevents parts from moving or falling out during machining — a common issue in school shops.

4. **Educational Licensing is Very Affordable** — MecSoft offers flexible options including:
    - Lab Licenses (network-restricted to one classroom)
    - Campus CAM (unlimited access across a school or district)
5. **AMS Provides Excellent Educator Support** — The Annual Maintenance Subscription includes access to the full CAMJam video library (dozens of hours of tutorials), monthly webinars, and direct technical support — extremely valuable for teachers who are not full-time CAM experts.
6. **Real Projects Increase Student Engagement** — Giving students the opportunity to design and machine functional items (furniture components, jewelry, signs, or molds) dramatically improves motivation and learning outcomes.
7. **Proof-of-Concept Machining is Recommended** — Many educators have students first machine their designs in inexpensive material (MDF, plywood, or foam) before cutting the final material. This teaches the importance of verification.

Key Takeaways

This webinar clearly demonstrates that MecSoft CAM products are well-suited for educational environments. The combination of an intuitive interface, powerful features (Knowledge Bases, Bridges/Tabs, Background Images), affordable educational licensing, and strong support resources makes it an excellent choice for high schools, technical colleges, and universities.

Biggest lessons:

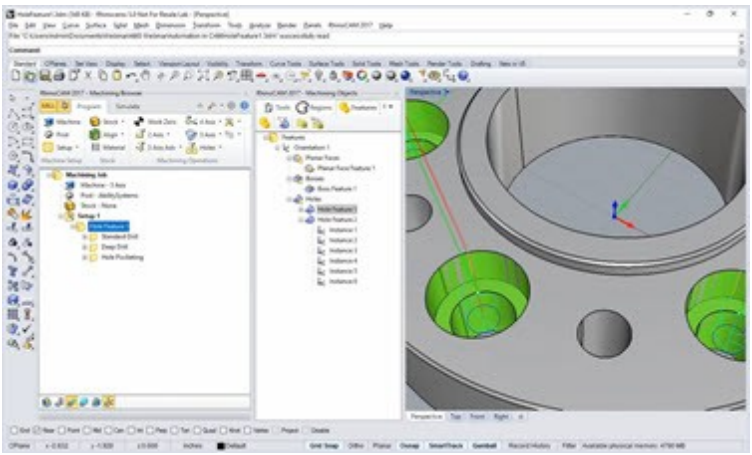
- The session proves that students can successfully produce professional-quality work when given the right tools and guidance.
- Real, meaningful student projects create high engagement.
- Features like Knowledge Bases and Bridges/Tabs reduce complexity for both teachers and students.
- Flexible licensing and excellent support (CAMJam + webinars) lower the barrier for schools.
- Start students with simple and achievable machining projects.
- Use standardized classroom workflows and reusable project templates.
- Teach simulation workflows before students operate CNC machines.
- Connect CAD design directly to physical manufacturing projects.
- Use reusable machining templates to simplify classroom instruction.
- Focus on hands-on CNC machining experience to reinforce learning.
- Organize classroom workflows to improve machine access and student productivity.

4.68 Automation in CAM

Link: https://www.youtube.com/watch?v=Ep3kfWvPbbc

This AMS webinar focuses on automation strategies inside CAM programming using RhinoCAM and VisualCAD/CAM. The presentation explains how CNC programmers and machinists can streamline repetitive programming tasks, improve workflow consistency, and reduce setup time using automation tools built into modern CAM systems. Topics include machining templates, automated toolpath generation, feature-based machining, setup automation, reusable machining databases, post-processing automation, and production efficiency improvements. Throughout the webinar, the presenters also share practical CAM automation techniques that help manufacturers improve scalability, reduce programming errors, and standardize machining operations across production environments.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Overview	Why automation matters and what will be covered
2:13	Knowledge Bases for Prismatic Parts	Saving operations (excluding geometry) and basic loading workflow
12:03	Knowledge Bases for 3-Axis Cavity/Core Parts	Applying the same KB to similar mold cavities
23:40	Feature-Based Machining Automation	Detecting features and using Whole Feature Knowledge Bases

Time	Topic / Section	Description
39:27	Creating Selection Rules	Color, layer, and geometry filters for automatic geometry selection
49:14	Saving Default Parameters	Creating custom defaults for operation types (faster than full KBs)
54:09	Saving CAM Preferences with Part Files	Ensuring consistent settings across computers
57:27	Conclusion & Best Practices Recap	Choosing the right automation method for your workflow

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Selection Rules = True Automation** — This is the most powerful feature. Create rules based on **color**, **layer**, or **geometry type** (e.g., "All red curves = pocketing", "All circles with diameter 6–10 mm = drilling"). When you load the Knowledge Base, the software automatically selects the matching geometry — you only need to click **Regenerate**.
2. **Whole Feature Machining is Extremely Efficient** — For prismatic parts with many identical features (e.g., multiple counter-bored holes), save operations to a **Whole Machining Knowledge Base**. The software can then automatically detect matching features and generate complete operation groups with one command.
3. **Use "Save as Defaults" for Simpler Automation** — When you only want to reuse parameters (not full operations), right-click an operation → **Save as Defaults**. Then set it as the active default in **CAM Preferences > Machining > Default Knowledge Base to Load**. This is faster than loading a full Knowledge Base for every new part.
4. **Enable "Show Load Settings from File Dialogue"** — In **User Interface Preferences**, turn this on. When you open a part file, the software will prompt you to load the CAM preferences (post settings, drill cycle behavior, etc.) saved with that file. This ensures consistency across different computers and users.
5. **Knowledge Bases Work Across All Axis Configurations** — The same automation techniques apply to 2-axis, 2.5-axis, 3-axis, 4-axis, and 5-axis operations.
6. **Unit Consistency is Critical** — Always verify that the units (inches vs millimeters) of the new part match the Knowledge Base you are loading. Mismatched units can cause incorrect depths and toolpaths.
7. **Combine Methods for Maximum Efficiency** — Use:

- **Selection Rules + Knowledge Bases** for parts with organized geometry (layers/colors)
- **Default Parameters** for quick parameter consistency
- **Whole Feature Machining** for highly repetitive prismatic features

8. **Start Simple** — Begin by saving just a few operations (e.g., one roughing + one finishing pass) to a Knowledge Base and test it on a similar part before building complex automated workflows.

Key Takeaways

This webinar is one of the **highest-ROI sessions** in the MecSoft AMS library. It shows how to move from manual, repetitive programming to true automation using **Knowledge Bases + Selection Rules**.

Biggest lessons:

- These automation techniques remain fully relevant and are still some of the most effective ways to scale productivity in current versions of RhinoCAM and VisualCAD/CAM.
- Selection Rules are the key to near-zero manual geometry selection.
- Whole Feature Machining dramatically speeds up prismatic parts.
- Default Parameters + file-based preferences ensure consistency across teams and machines.
- Build reusable machining templates to reduce programming time.
- Standardize feeds, speeds, and tooling databases for consistency.
- Use feature-based machining to automate operation assignment.
- Organize reusable operation libraries for efficient CAM workflows.
- Apply automation strategically to repeat production jobs.
- Improve programming consistency across multiple operators.
- Validate automated machining operations using simulation tools.

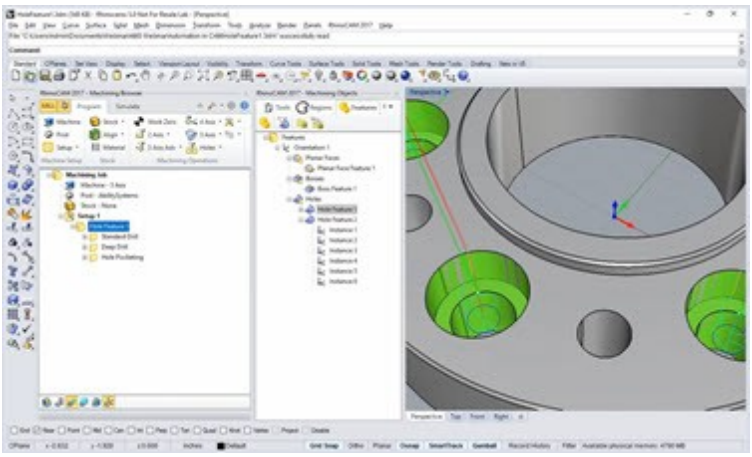
4.69 Feature Machining

Link: https://www.youtube.com/watch?v=hxo0c8iEuaA

This AMS webinar focuses on Feature Detection Machining workflows inside RhinoCAM and VisualCAD/CAM. The presentation explains how CAM software can automatically recognize machinable features from solid models and apply machining operations with minimal manual input. Topics include automatic feature recognition, hole detection, pocket recognition, machining automation, operation templates, tool selection, feature editing, setup workflows, simulation, and post processing. Throughout the webinar, the presenters also share practical machining recommendations for improving programming efficiency, reducing repetitive CAM

tasks, and streamlining CNC production workflows using automated feature-based machining strategies.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction by Anita Non	Webinar goals and overview of Feature Detection Machining
1:40	Prismatic Feature Detection in Visual CAM	Detecting pockets, holes, and planar faces; creating operations from selected faces
8:25	Feature Detection in RhinoCAM	Single-button detection, automatic hole grouping by diameter/depth
19:00	Complex Prismatic Parts in Visual CAM for SolidWorks	Multi-level pockets, hole collectors, counter-bores, and hole profiling
25:20	Advanced Prismatic Features (Bosses and Slots)	New Slotting operation with intelligent high-speed vs center-line patterns
33:45	Setting Work Zero for Multiple Orientations	Rotating coordinate systems and defining Work Zero per setup
36:30	Non-Prismatic Feature Detection	Mold cavity example — detecting prismatic sub-features and applying 3-axis operations

Time	Topic / Section	Description
41:30	Conclusion and Q&A	Benefits, configuration availability, and final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Hole Feature Collectors Are Extremely Powerful** — Holes with identical diameter and depth are automatically grouped. You can apply drilling, spot drilling, or profiling operations to the entire group in one step.
2. **The Slotting Operation is Smart** — When the tool diameter is smaller than the slot width, it automatically generates high-speed (trichoidal) cut patterns. When the tool matches the slot width, it creates a simple center-line pass.
3. **Multi-Orientation Detection Creates Setups Automatically** — When you detect features from different sides (top, bottom, side), the software automatically creates new Setups and coordinate systems for each orientation.
4. **Feature Detection Works on Both Prismatic and Non-Prismatic Parts** — Even on complex mold cavities, it can identify prismatic sub-features (small pockets, holes) and suggest appropriate 3-axis operations (Horizontal Roughing, 3D Offset Pocketing, etc.).
5. **Boundaries and Depths Are Automatically Derived** — Once a feature is detected, the software automatically calculates the correct machining boundaries and depths from the geometry — eliminating most manual region selection.
6. **Work Zero Can Be Set Relative to Stock or Part** — For each new orientation/setup, you can define the Work Zero based on the stock box or specific part geometry for that side.
7. **Feature Detection is Available Across Most Configurations** — It works in Standard, Expert, Professional, and Premium versions (with increasing capability in higher configurations).
8. **Traditional Region Selection Still Available** — Feature Detection is a major time-saver, but you can still fall back to manual region selection for fine control or non-standard geometry.

Key Takeaways

This webinar introduced one of the biggest productivity improvements: Feature Detection Machining. It allows users to go from a solid model to complete toolpaths with dramatically less manual geometry selection, especially on prismatic and semi-prismatic parts.

Biggest value highlighted:

- The techniques shown remain foundational and are still widely used in current versions of RhinoCAM and VisualCAD/CAM.
- Automatic hole grouping and batch operations
- New intelligent Slotting operation
- Multi-orientation support with automatic Setup creation
- Works on both prismatic parts and complex 3D geometries (mold cavities)
- Use clean solid models to improve feature recognition accuracy.
- Organize CAD geometry consistently for efficient feature detection workflows.
- Build reusable machining templates to reduce programming time.
- Verify automatically assigned machining operations before posting code.
- Use feature-based machining for repeat production jobs and similar part families.
- Combine automation with manual optimization for the best machining results.
- Validate toolpaths and machining motion using simulation tools.

4.70 CAM in Education

Link: https://www.youtube.com/watch?v=6eczE0zusZs

This MecSoft webinar focuses on CAM education and how RhinoCAM and VisualCAD/CAM are used in academic environments, training programs, and technical manufacturing courses. The presentation explains how educational institutions can integrate CAM software into machining curricula to teach CNC programming, manufacturing fundamentals, and real-world machining workflows. Topics include CAD/CAM integration, student learning workflows, machine simulation, toolpath creation, educational licensing, project-based learning, CNC machining fundamentals, and industry preparation. Throughout the webinar, the presenters also share practical recommendations for instructors, students, and educational programs seeking to improve manufacturing education using modern CAM technologies.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and overview of CAM in education
3:00	Why MecSoft CAM for Schools	Ease of use, affordability, and industry relevance
8:30	Guest Educator Presentations	Real classroom experiences and curriculum examples
15:00	Student Project Showcase	Examples of student work (signs, furniture components, molds, jewelry)
22:00	Live Toolpath Demonstration	RhinoCAM or VisualCAD/CAM in action on student geometry
30:00	Educational Licensing Options	Lab Licenses, Campus CAM, instructor keys, and pricing
37:00	AMS Benefits for Educators	Access to CAMJam video library, monthly webinars, and support
42:00	Q&A Session	Common questions from teachers and administrators
50:00	Conclusion & Resources	Final thoughts and links to educational materials

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the webinar:

1. **Start with 2.5-Axis Projects** — Many educators begin with signs, plaques, and simple 2.5-axis parts before moving to full 3-axis work. This builds student confidence quickly.
2. **Use Background Bitmaps for Tracing** — When students work from logos or hand-drawn designs, import the image as a **Background Bitmap** in VisualCAD/CAM or Rhino. This makes accurate curve tracing much easier.
3. **Knowledge Bases Save Teaching Time** — Teachers can create and save **Knowledge Bases** with pre-configured tools, tolerances, and strategies. Students can load the correct KB and focus on geometry selection rather than parameter setup.

4. **Bridges/Tabs Prevent Lost Parts** — Always use the **Bridges/Tabs** feature when cutting multiple parts from one sheet. This is especially important in school shops where parts can move or fall out during cutting.
5. **Educational Licensing is Very Affordable** — MecSoft offers flexible options including **Lab Licenses** (network-restricted to one classroom) and **Campus CAM** (unlimited access across a school or district).
6. **AMS Provides Strong Educator Support** — The Annual Maintenance Subscription includes access to the full **CAMJam** video library and monthly webinars — extremely valuable for instructors who are not full-time CAM experts.
7. **Real Projects Increase Engagement** — Giving students the opportunity to design and machine functional items (furniture, signs, jewelry, or molds) dramatically improves motivation and learning outcomes.
8. **Proof-of-Concept Machining is Recommended** — Many educators have students first machine their designs in inexpensive material (MDF or plywood) before cutting the final material. This teaches the importance of verification.

Key Takeaways

This webinar is an excellent early resource showing that **MecSoft CAM products are well-suited for educational environments**. The combination of an intuitive interface, powerful productivity features (Knowledge Bases, Bridges/Tabs, Background Bitmaps), affordable educational licensing, and strong support resources makes it an excellent choice for schools.

Biggest lessons:

- Real, meaningful student projects create high engagement.
- Features like Knowledge Bases and Bridges/Tabs reduce complexity for both teachers and students.
- Flexible licensing and excellent support (CAMJam + webinars) lower the barrier for schools.
- Integrate CAD and CAM instruction together for better workflow understanding.
- Use simulation tools extensively to reinforce machining safety and visualization.
- Begin with simple machining exercises before introducing advanced operations.
- Teach real-world manufacturing workflows and setup procedures early.
- Create reusable machining templates for classroom efficiency.
- Combine CAM instruction with hands-on CNC machine experience whenever possible.
- Emphasize process planning and machining verification techniques.

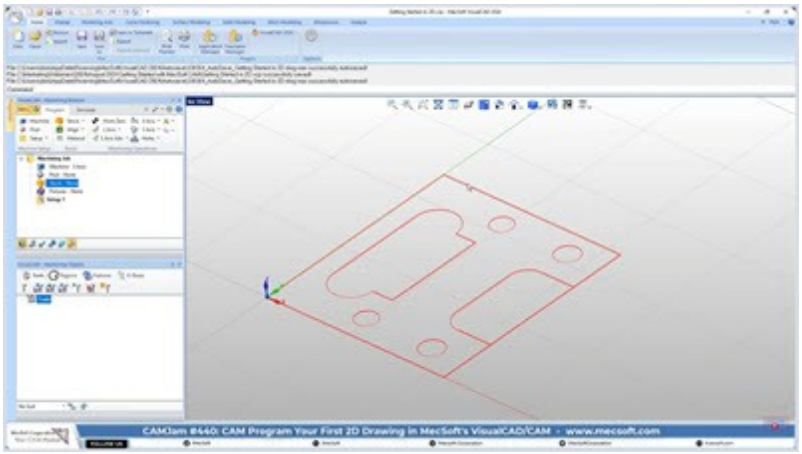
VisualCAD Videos

5.1 VisualCAD Shorts

Link: Select a Video Link from the List Below.

This collection of short tutorials is a combination of over 1.5 hours of training and teaches essential VisualCAD skills — from basic curve editing and modeling to geometry repair, masking, dimensioning, and toolpath preparation — helping users efficiently turn imported drawings into clean, machinable geometry.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
1:46	Dynamic Dimensions	Demonstrates the Dynamic Dimensions feature in VisualCAD
5:09 2:12	Using Background Images 1 Using Background Images 2	How to use Background Images as visual aids
1:19	Offset & Fillet Corner Options	Demonstrates the Offset and Fillet corner options in VisualCAD
1:59 3:53	How to Orient a Part 1 How to Orient a Part 2	How to quickly orient a 3D part in VisualCAD/CAM using the “Orient Part” command

Time	Topic / Section	Description
1:31	VisualCAD Feature Presentation	showcases key modeling tools, 2D/3D geometry creation, editing, surface modeling, and overall workflow advantages
2:43	How to Model Solid Extrusions	How to create 3D solid models from 2D drawings in VisualCAD
3:36	How to Check the Units	How to check and change units (inches vs millimeters) in VisualCAD
2:28	Working with AutoCAD Files	How to import and work with AutoCAD files (DXF/DWG) in VisualCAD
2:26	How to Use the Transform Menu	How to use the Transform Menu in VisualCAD to move, rotate, scale, mirror, and array selected geometry
1:29	How and Why to Merge Curves	Why and how to merge curves in VisualCAD. After importing DXF files
4:42	Drawing Rectangular Arrays	How to create Rectangular Arrays in VisualCAD. It shows selecting objects, using the Array command
8:51	How to Convert RhinoCAM Toolpaths into VisualCAD/CAM	demonstrates how to seamlessly transfer complete machining jobs from RhinoCAM to VisualCAD/CAM using the Export the VCP utility
3:36	How to Analyze Geometry	How to use the Analyze menu in VisualCAD to inspect imported or created geometry. It demonstrates measuring arc/circle diameters, checking overall part dimensions
3:09	How to Section a 3D Model	How to create 2D sections through a 3D model in VisualCAD using the Section command
2:04	Basic Curve Filleting	How to select multiple curves, apply a fillet radius, and create smooth rounded corners
1:49	Trimming Curves	Shows selecting two intersecting curves, using the Trim command, and choosing which segments to keep or

Time	Topic / Section	Description
		remove
3:19	Basic Curve Editing	Covers trimming intersecting curves, using the Continuous Command toggle for repeated operations, merging multiple curve segments into single continuous entities and more
3:46	Basic Drawing	How to Explode and Join curves, breaking complex curves into individual segments using Explode, then re-joining selected segments
4:48	Understanding VisualCAD Options	How to access and understand the Options menu in VisualCAD
5:12	Masking Holes	How to create basic planar surfaces from two corner points or from surface edges
1:45	Analyze Distances	How to analyze distance between objects and to easily move geometry using the Graphics Manipulator
5:10	Surface Modeling	Basic Surface Modeling, how to create surfaces from curve geometry, how to build clean 3D surfaces from 2D curves and more
2:44	Extrude a Solid	How to extrude curves into solid models in VisualCAD

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from this tutorial:

1. **Dynamic Dimensions in VisualCAD** — This short tutorial demonstrates the Dynamic Dimensions feature in VisualCAD. As you measure distances between points or analyze diameters from solid model edges/arcs, the values display live on-screen in real time.
2. **Using Background Images as Visual Aids in VisualCAD** — This short tutorial demonstrates how to use Background Images as visual aids in VisualCAD. It shows how to insert bitmap images (JPG, PNG, etc.), scale and position them, adjust transparency/gray-scale, and use them as references to accurately trace curves and create geometry with VisualCAD's drawing and modeling tools.
3. **Offset & Fillet Corner Options in VisualCAD** — This short tutorial demonstrates the Offset and Fillet corner options in VisualCAD. Users can now choose Sharp, Round (default), or

Fillet (with zero radius) when offsetting curves. The video shows how to create sharp corner offsets and use zero-radius fillets to extend and intersect open curves cleanly.

4. **How to Orient a Part in VisualCADCAM** — This short tutorial demonstrates how to quickly orient a 3D part in VisualCAD/CAM using the "Orient Part" command. It shows how to align any face of the model to the top (positive Z) for proper machining setup, including rotating the part and stock together while preserving the world coordinate system.

5. **VisualCAD Feature Presentation** — This video provides a clear feature presentation of VisualCAD, the powerful CAD component of VisualCAD/CAM. It showcases key modeling tools, 2D/3D geometry creation, editing capabilities, surface modeling, and overall workflow advantages for users who need a dedicated, shop-friendly CAD environment paired with integrated CAM.

6. **How to Model Solid Extrusions in VisualCAD** — This short tutorial demonstrates how to create 3D solid models from 2D drawings in VisualCAD using the Solid Modeling menu. It covers extruding closed curves into solids, managing layers, and basic modeling workflows.

7. **How to Check the Units in VisualCAD** — This short tutorial demonstrates how to check and change units (inches vs millimeters) in VisualCAD. It shows where to find the current units display, how to verify model scale after import, and the proper way to convert units using the Scale command or unit settings dialog for accurate machining.

8. **Working with AutoCAD Files in VisualCAD** — This tutorial demonstrates how to import and work with AutoCAD files (DXF/DWG) in VisualCAD. It shows opening 2D drawings and 3D models, fixing common issues like scale and color, basic drawing edits, curve merging, and preparing geometry for CAM toolpaths. Ideal for users transitioning from AutoCAD to VisualCAD/CAM.

9. **How to Use the Transform Menu in VisualCAD** — This short tutorial demonstrates how to use the Transform Menu in VisualCAD. It shows how to move, rotate, scale, mirror, and array selected geometry. It explains the basic options and emphasizes that transformations apply only to currently selected objects, making it a fundamental tool for positioning and editing parts before CAM programming.

10. **How and Why to Merge Curves in VisualCAD** — This short tutorial explains why and how to merge curves in VisualCAD. After importing DXF files, geometry often arrives as many small segments. It demonstrates selecting all curves and using the Merge command to create single, continuous closed curves — essential for clean pocketing, profiling, and reliable toolpath generation.

11. **Drawing Rectangular Arrays in VisualCAD** — This short tutorial demonstrates how to create Rectangular Arrays in VisualCAD. It shows selecting objects, using the Array command, specifying rows/columns, spacing distances, and previewing the result. It emphasizes calculating distances ahead of time to accurately fill sheets or layouts for efficient machining preparation.

12. **How to Convert RhinoCAM Toolpaths into VisualCAD/CAM** — This short tutorial demonstrates how to seamlessly transfer complete machining jobs from RhinoCAM to VisualCAD/CAM using the Export the VCP utility. It shows exporting a full job (geometry + toolpaths) as a native .VCP file, then opening it in VisualCAD/CAM with perfect data integrity — ideal for engineering-to-shop-floor workflows.

13. **How to Analyze Geometry in VisualCAD** — This short tutorial shows how to use the Analyze menu in VisualCAD to inspect imported or created geometry. It demonstrates measuring arc/circle diameters, checking overall part dimensions (length/width), and getting quick insights into drawing size and scale — essential when working with files from other CAD systems.

14. **How to Section a 3D Model in VisualCAD** — This short tutorial demonstrates how to create 2D sections through a 3D model in VisualCAD using the Section command. It shows selecting a plane (e.g., XY), adjusting the section location, and generating clean cross-section curves for better visualization, measurement, and 2D machining preparation of complex 3D parts.

15. **Basic Curve Filleting in VisualCAD** — This short tutorial demonstrates Basic Curve Filleting in VisualCAD. It shows how to select multiple curves, apply a fillet radius, and create smooth rounded corners. It explains the command options and highlights how filleting improves toolpath quality and part aesthetics for machining.

16. **Trimming Curves in VisualCAD** — This short tutorial demonstrates how to trim curves in VisualCAD. It shows selecting two intersecting curves, using the Trim command, and choosing which segments to keep or remove. It also explains the "Continuous Command" toggle for repeated trimming and the importance of curves being on the same plane.

17. **Basic Curve Editing in VisualCAD** — This short tutorial demonstrates basic curve editing in VisualCAD. It covers trimming intersecting curves, using the Continuous Command toggle for repeated operations, merging multiple curve segments into single continuous entities, and Explode for breaking curves apart. These tools are essential for cleaning imported geometry before creating accurate toolpaths.

18. **Basic Drawing in VisualCAD** — This short tutorial demonstrates how to Explode and Join curves in VisualCAD. It shows breaking complex curves into individual segments using Explode, then re-joining selected segments into single continuous curves with the Join command. These tools are essential for cleaning up imported geometry before creating reliable toolpaths.

19. **Understanding VisualCAD Options** — This short tutorial explains how to access and understand the Options menu in VisualCAD. It covers key settings such as modeling units (inches vs mm), display preferences, tolerance controls, and other default behaviors that affect modeling accuracy and CAM performance. It emphasizes checking these options when starting new projects.

20. **Masking Holes in VisualCAD** — This short tutorial demonstrates how to create basic planar surfaces from two corner points or from surface edges. It shows placing them on a dedicated red layer, and toggling visibility to control whether toolpaths (like parallel finishing) enter or avoid the holes.
21. **Analyze Distances in VisualCAD** — This short tutorial demonstrates how to analyze the distance between objects in VisualCAD as well as using those distance values to easily move geometry using the Graphics Manipulator.
22. **Surface Modeling in VisualCAD** — This short tutorial demonstrates basic Surface Modeling in VisualCAD. It shows how to create surfaces from curve geometry using commands like Surface of Extrusion and other surface tools. It explains how to build clean 3D surfaces from 2D curves for better visualization and CAM toolpath preparation.
23. **Extrude a Solid in VisualCAD** — This short tutorial demonstrates how to extrude curves into solid models in VisualCAD. It shows selecting closed curves, using the Extrude command from the Solid Modeling menu, setting extrusion height/direction, and creating clean, high-integrity solids ready for CAM toolpaths.

Key Takeaways

This collection of short tutorials teaches essential VisualCAD skills — from basic curve editing and modeling to geometry repair, masking, dimensioning, and toolpath preparation — helping users efficiently turn imported drawings into clean, machinable geometry.

Best Practices:

- Dynamic Dimensions display live measurements for quick geometry analysis.
- Background Images serve as accurate visual references for tracing and modeling.
- Offset and Fillet options give precise control when modifying curves.
- Orient Part command quickly aligns 3D models for proper machining setup.
- VisualCAD is a dedicated CAD environment optimized for integrated CAM workflows.
- Solid Extrusions convert 2D closed curves into high-integrity 3D solids.
- Checking and converting units prevents scaling errors after file imports.
- AutoCAD (DXF/DWG) files can be imported, cleaned, and prepared for CAM.
- Transform Menu allows easy moving, rotating, scaling, and arraying of objects.
- Merging curves creates continuous profiles essential for reliable toolpaths.
- Rectangular Arrays efficiently duplicate geometry for sheet layouts and patterns.
- RhinoCAM toolpaths export as native .VCP files for seamless VisualCAD transfer.

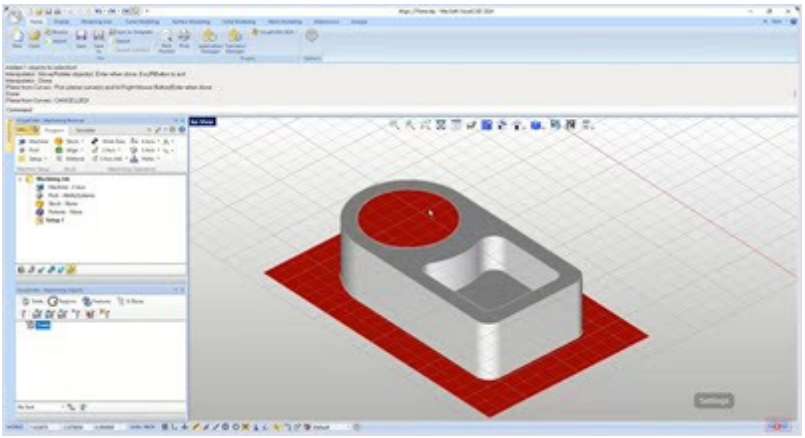
- Analyze menu quickly measures distances, diameters, and part dimensions.
- Section command generates clean 2D cross-sections from 3D models.
- Basic Curve Filleting creates smooth rounded corners for better toolpaths.
- Trimming curves removes unwanted segments at intersections.
- Basic Curve Editing (Explode, Join, Merge) cleans imported geometry.
- Options menu controls units, display, tolerance, and default behaviors.
- Masking surfaces protect holes and features during toolpath generation.
- Surface Modeling builds 3D surfaces from curves for visualization and machining.
- Extruding solids from closed curves creates ready-to-machine 3D models.

5.2 Using VisualCAD

Link: https://www.youtube.com/watch?v=tWY6tm5nA-o

This is a foundational “getting started” webinar for VisualCAD/CAM. It covers the key differences from Rhino, file conversion/import workflows, basic modeling tools, and practical 2-axis & 3-axis CAM programming — ideal for new users or shop-floor teams transitioning to VisualCAD/CAM.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals, chat/Q&A, video distribution info

Time	Topic / Section	Description
2:30	Differences Between Rhino and VisualCAD	Shop-floor focus vs advanced engineering modeling
7:30	Conversions to VisualCAD & Translators	RhinoCAM/SolidWorks export (.VCP), STEP/IGES import, orientation tools
14:00	Transform 2D Drawings & Measure/Analyze Geometry	DXF import, curve merging, measurement tools, stock bounding box
25:00	Modeling from DXF Files & Basic Modeling	Extrude curves, surface creation, trimming, filleting, section curves
35:00	VisualCAD Options	System/display settings, tolerance controls, grid & view preferences
38:00	2-Axis CAM Programming	DXF import → pocketing & profiling with ramp entry
50:00	3-Axis CAM Programming	STEP import, horizontal roughing, pocketing, profiling, spiral finishing
1:05:00	Resources & Support	Online help, VisualSERVE portal, AMS CAMJam videos, technical support

Key Tips & Best Practices Extracted

Here are the most practical takeaways from the webinar:

1. **VisualCAD is Shop-Floor Focused** — Use it for simpler geometry (DXF files, 2D machining, laser/plasma) rather than complex parametric engineering work. Rhino is better for advanced modeling and automation.
2. **Native .VCP Export from RhinoCAM/SolidWorks** — Use the built-in export utility to transfer complete jobs (geometry + toolpaths) into VisualCAD with zero data loss.
3. **Merge Curves Early** — Always merge curves after importing DXF files before extruding or creating toolpaths for clean, continuous profiles.
4. Use Background Images or Section Curves — When working from 2D drawings, import elevation views or create section curves to accurately determine depths and internal features.
5. **Stock Bounding Box Tool** — Quickly determine part size and verify dimensions before programming.
6. **Align Stock to Top of Geometry** — Set Z=0 at the top of the part for consistent and intuitive depth management.

7. **Ramp Entry/Exit in 2-Axis Operations** — Use ramp entry instead of plunging when geometry allows, especially on aluminum or harder materials.
8. **Predefined Regions for Efficiency** — Create and save named regions for repeated features (pockets, hole groups) to speed up programming.
9. **Tight Tolerances for Accuracy** — Use tight geometry tolerance settings during import and modeling; loosen curve hookup tolerance only if merging fails.
10. **Layer Management** — Organize geometry on separate layers (e.g., red for visibility) to keep complex jobs manageable.
11. **Simulate After Every Operation** — Always run cut material simulation to verify toolpaths, especially when switching between 2-axis and 3-axis strategies.
12. **Orientation Utility** — After importing STEP/IGES files, use the orientation tool to quickly align the part with Z upward for standard machining setup.
13. **Spiral Finishing for Better Surface Finish** — Use spiral finishing with small step-over (e.g., 0.1") and a ball mill for final passes on contoured areas.

Key Takeaways

This webinar is the ideal starting point for anyone new to VisualCAD/CAM. It clearly explains:

- When to choose VisualCAD vs Rhino
- How to efficiently import and prepare 2D drawings and 3D models
- Practical 2-axis and 3-axis workflows from real examples

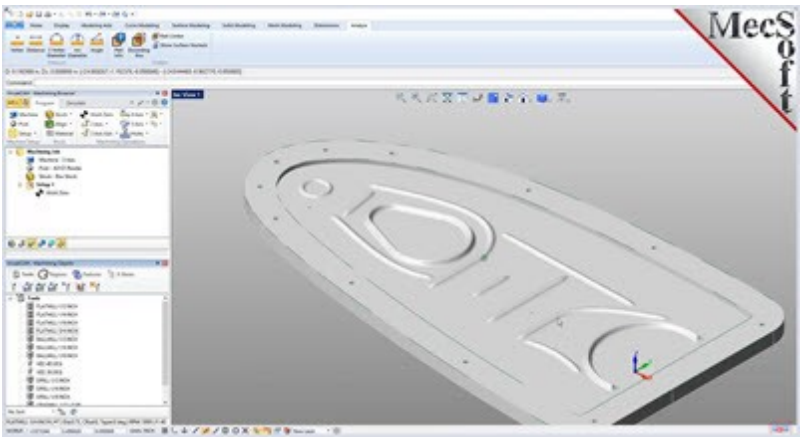
It also highlights the strong interoperability between RhinoCAM and VisualCAD/CAM — a major advantage for companies with both engineering and shop-floor teams.

5.3 Using VisualCAD

Link: https://www.youtube.com/watch?v=x7Q4F_2uPqI

This CAMJam episode provides a clear, step-by-step introduction to working with VisualCAD (the CAD environment) and VisualMILL (the CAM module) together. It is designed for new users who want to understand the full workflow from geometry creation/import to finished G-code.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Overview	Purpose of the episode and what will be covered
2:15	VisualCAD Interface Overview	Main windows, toolbars, layers, and construction plane
7:40	Creating Basic Geometry in VisualCAD	Drawing rectangles, circles, and using trim/extend
13:20	Importing Geometry	Importing DXF, DWG, and STEP files
17:50	Stock Definition & Work Zero	Setting up stock and defining the machining origin
22:30	Switching to VisualMILL	How to access the CAM module and Machining Browser
26:10	2.5-Axis Toolpath Creation	Facing, pocketing, and profiling operations
33:45	Simulation & Verification	Running cut material simulation and checking for errors
38:20	Post-Processing	Generating G-code and understanding post settings
42:00	Tips, Best Practices & Q&A	Common questions and workflow recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from this CAMJam episode:

1. **Work in VisualCAD First, Then Switch to VisualMILL** — Always create or import your geometry in VisualCAD before switching to the CAM module. This keeps your CAD and CAM work cleanly separated.
2. **Use Layers Effectively** — Organize geometry on separate layers (e.g., “Stock”, “Part”, “Holes”, “Profiles”). You can then easily turn layers on/off when selecting geometry for operations.
3. **Set Work Zero at the Top of Stock** — This is the most intuitive and reliable convention. It makes all depth values positive numbers going downward from zero.
4. **Merge Curves Before Creating Toolpaths** — Always merge multiple curve segments into single continuous entities before using them for pocketing or profiling. This prevents gaps and broken toolpaths.
5. **Start with Simple Operations** — For new users, begin with Facing → Pocketing → Profiling. Master these three operations before moving to more advanced strategies.
6. **Always Simulate Before Posting** — Run cut material simulation after every major operation (especially roughing) to verify that the toolpath behaves as expected.
7. **Use Ramp Entry for Aluminum and Harder Materials** — Avoid plunging directly into material. Use ramp or helical entry whenever possible to reduce tool stress.
8. **Predefined Regions Save Time** — Create and save named regions for common features (e.g., “All 1/4" Holes”). This makes selecting geometry much faster on future jobs.
9. **Stock Bounding Box is Your Friend** — Use the Stock Bounding Box tool to quickly define stock that exactly matches your part’s overall size.
10. **Keep the Machining Browser Visible** — Dock the Machining Browser on the side of the screen so you can easily see your operations tree while working in VisualCAD.

Key Takeaways

This CAMJam episode is an excellent starting point for anyone new to the VisualCAD/CAM suite. It clearly shows the natural workflow:

VisualCAD (Design/Import) → VisualMILL (Toolpaths & Simulation) → Post-Processing

The biggest lessons are:

- Keep your CAD work clean and organized (layers, merged curves).
- Set Work Zero at the top of stock.
- Simulate frequently.
- Start simple and build confidence with basic operations.

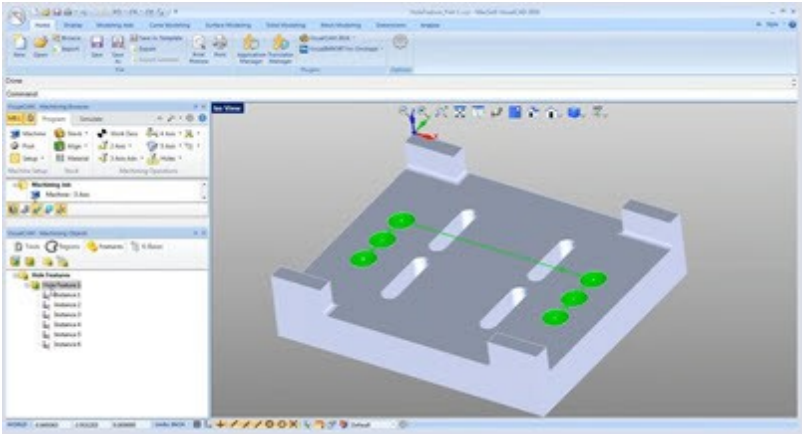
This video is ideal for beginners who want to understand how the CAD and CAM parts of the software work together.

5.4 Using VisualCAD

Link: https://www.youtube.com/watch?v=pbZjX9fir8o

This webinar introduces the VisualCAD/CAM suite, covering the four main modules (VisualMill, VisualTurn, VisualNest, VisualArt), new features and practical demonstrations — especially the powerful new Automatic Feature Detection & Machining and Knowledge Base Automation capabilities.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Housekeeping	Webinar goals and product overview
4:00	VisualCAD/CAM Product Overview	Four modules (Mill, Turn, Nest, Art), file translators, and Onshape integration
11:20	VisualMill Interface & Workflow	Machining Browser, Tools/Regions/Feature Detection tabs, and basic setup
16:00	2.5-Axis Programming Demo	Facing, pocketing, and the new 2-Axis Roughing operation
24:00	3-Axis Programming Enhancements	Parallel finishing improvements and “Tops Only / Tops and Sides” options
32:00	4-Axis & 5-Axis Overview	Indexed and continuous multi-axis methods
39:00	New: Whole Feature Detection & Machining	Automatic detection of holes and creation of complete operation sets (spot drill → drill → profile)

Time	Topic / Section	Description
44:00	New: Knowledge Base Automation with Selection Rules	Assigning rules (by layer/color) so operations automatically select geometry on new parts
50:00	New: Explode Cabinet Design Utility	Automatically converts 3D cabinet models into flat 2D panels for nesting
55:00	VisualNest & VisualArt Demonstrations	Rectangular nesting of exploded cabinet panels and converting artwork to 3D reliefs
1:00:00	Q&A and Closing Remarks	SolidWorks plug-in status, post-processor customization, and Onshape integration

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from this 2016 webinar:

1. **Automatic Feature Detection + Knowledge Bases = Major Time Saver** — Even in its first version, this combination allowed users to program prismatic parts with repetitive features (especially holes) dramatically faster by detecting features and automatically applying complete operation sets.
2. **Selection Rules Make Automation Practical** — You can assign rules (e.g., "all curves on Layer 5 = pocketing") so that when you load a Knowledge Base on a new part, the software automatically selects the correct geometry.
3. **Explode Cabinet Design is Extremely Useful for Cabinet Shops** — This utility automatically breaks down 3D cabinet assemblies into individual flat panels (with holes grouped to their parent parts), ready for nesting — a huge time saver.
4. **2-Axis Roughing** — Combines facing and roughing into one intelligent operation that automatically determines what needs to be cut based on part boundaries and stock.
5. **Parallel Finishing "Tops Only" Option** — A very useful new setting that machines only the top surfaces, avoiding unnecessary side machining on certain parts.
6. **Knowledge Bases Work Across Configurations** — The automation features (Feature Detection + Selection Rules) were available across multiple configurations, not just the highest ones.

Key Takeaways

This 2016 webinar introduced Automatic Feature Detection & Machining and Knowledge Base Automation with Selection Rules — two foundational technologies that would become even more powerful in subsequent years (especially 2018+).

Biggest highlights of the 2016 release:

- First version of Automatic Feature Detection + Whole Feature Machining

- Knowledge Base automation with selection rules
- Explode Cabinet Design utility (very popular with cabinet makers)
- Onshape cloud integration
- Many usability and performance improvements

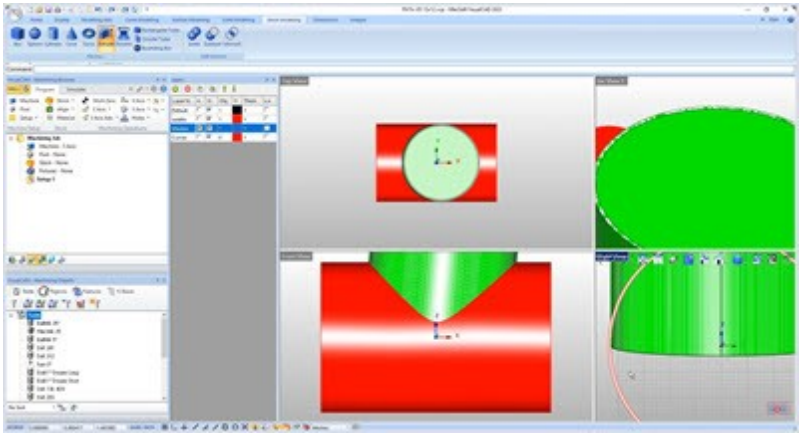
This release marked the beginning of MecSoft's strong push toward intelligent automation, which has continued to evolve in later versions.

5.5 Cut Tubular Intersections

Link: https://youtu.be/ozJ6GK7wOtg

This is a concise, practical tutorial showing how to machine the intersection curve between two circular pipes (tubular joints) in VisualCAD/CAM. It demonstrates a clever mesh-based workflow to generate the exact intersection profile when solid Boolean operations are difficult.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Problem Overview	Why tubular intersections are challenging and the mesh-based solution
1:30	Orient Construction Plane	Using "Set Plane to a Face" to align the CPlane normal to the pipe
3:00	Create Mesh Tubes	Extruding outer edge curves to create mesh representations of both pipes
5:00	Mesh Subtraction (Boolean Difference)	Subtracting one mesh from the other to isolate the intersection edge curve

Time	Topic / Section	Description
7:00	Extract Silhouette Curve	Generating a clean, machinable curve from the intersection edge
8:30	Curve Cleanup (Explode & Merge)	Breaking and re-merging curve segments into a single continuous profile
10:00	2-Axis Profiling Setup	Creating the profiling operation, stock definition, and toolpath generation
11:30	Final Toolpath & Simulation	Verifying the cut on the tubular intersection

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from this short tutorial:

1. **Use Meshes Instead of Solids for Complex Intersections** — When solid Boolean operations fail or produce poor results on tubular parts, convert both pipes to meshes. Mesh subtraction is often more reliable for finding the exact intersection curve.
2. **Orient the Construction Plane First** — Use Set Plane to a Face (or equivalent) to make the CPlane perpendicular to the main pipe's axis. This ensures clean extrusion and accurate intersection geometry.
3. **Extrude Outer Edge Curves** — Create mesh tubes by extruding the outer edge curves of each pipe rather than trying to use the solid bodies directly.
4. **Mesh Subtraction = Intersection Curve** — Subtracting one mesh from the other cleanly reveals the intersection edge curve that would be extremely difficult to create manually.
5. **Silhouette Curve is the Machinable Profile** — After getting the raw intersection edge, extract a Silhouette Curve. This gives you a clean, single continuous curve suitable for 2-axis profiling.
6. **Always Merge Curves Before Profiling** — The silhouette extraction often produces multiple small segments. Explode and then merge them into one continuous curve for a smooth, uninterrupted toolpath.
7. **Simple 2-Axis Profiling is Sufficient** — Once you have a clean intersection curve, a standard 2-axis profiling operation with proper containment and stock definition is all that's needed.
8. **Cylinder Stock from Bounding Box** — Define stock as a cylinder using the model bounding box for quick and accurate stock representation.

Key Takeaways

This short tutorial provides a clever, reliable workflow for machining tubular intersections — a common challenge in fabrication, piping, and structural work.

The core technique:

1. Orient the construction plane correctly
2. Create mesh representations of both pipes
3. Use mesh Boolean subtraction to find the intersection
4. Extract a clean silhouette curve
5. Merge the curve and profile it with a standard 2-axis operation

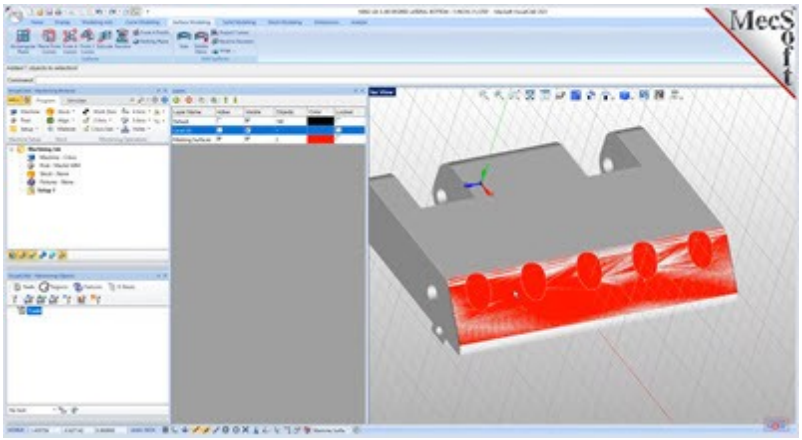
This mesh-based method is often faster and more robust than trying to create the intersection curve with pure solid modeling.

5.6 Model Masking Surfaces

Link: https://youtu.be/MVCl-kbCtd0

This is a short, highly practical tutorial that shows a fast and effective method for creating masking surfaces in VisualCAD/CAM. These temporary surfaces are used to prevent toolpaths from entering protected areas (such as holes) during roughing or finishing operations.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:11	Introduction & Purpose of Masking Surfaces	Why masking surfaces are useful (protecting holes during parallel finishing)
0:42	Creating Edge Curves	Selecting model edges and generating edge curves
1:02	Surface of Extrusion	Creating planar masking surfaces by extruding edge curves

Time	Topic / Section	Description
1:33	Organizing Surfaces on a Separate Layer	Creating a dedicated “masking surfaces” layer and coloring it red
2:48	Using Masking Surfaces in CAM Operations	How VisualCAD/CAM respects visible masking surfaces and how to toggle them on/off

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from this short tutorial:

1. **Masking Surfaces Prevent Tool Entry** — Use them to stop the tool from machining into holes, pockets, or other areas you want to protect during roughing or finishing passes.
2. **Extrude Edge Curves for Fast Planar Surfaces** — The quickest way to create masking surfaces is to:
   - Create edge curves from the model
   - Use Surface of Extrusion
   - Snap the extrusion to the opposite edge
3. **Use a Dedicated Red Layer** — Create a separate layer called “masking surfaces” and set it to red. This makes the surfaces highly visible and easy to toggle on/off with a single click.
4. **Toggle Layer Visibility to Control Toolpaths** — When the masking surfaces layer is on, the toolpath will respect the surfaces and avoid those areas. When you turn the layer off, the toolpath will machine through those areas.
5. **Keep Masking Surfaces Simple** — Planar or lightly extruded surfaces work best. Avoid overly complex masking geometry unless absolutely necessary.
6. **Hide Stock When Using Masking Surfaces** — It is often helpful to hide the stock model so the toolpath can extend beyond the part boundaries when needed.
7. **Masking Surfaces Work with Most 3-Axis Operations** — They are especially effective with Parallel Finishing, Horizontal Finishing, and Horizontal Roughing.

Key Takeaways

This short tutorial demonstrates one of the most useful techniques in MecSoft CAM for controlling toolpath behavior without manually creating complex containment curves.

The core workflow is simple and fast:

- Create edge curves
- Extrude planar masking surfaces
- Put them on a red “masking surfaces” layer
- Toggle the layer on/off to control whether the toolpath respects the protected areas

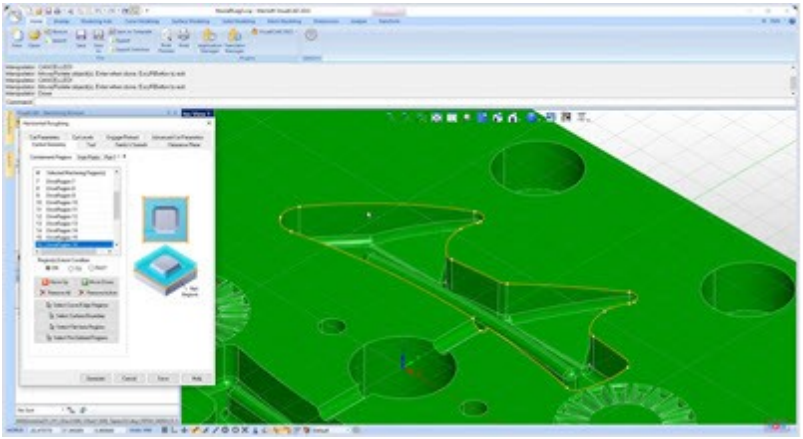
This method is especially valuable when working with parts that have many holes or features you want to protect during roughing but machine later.

5.7 Repair Control Geometry

Link: https://youtu.be/W6OftTjnLvg

This is a **short, focused tutorial** that addresses one of the most common errors in MecSoft CAM: **"Open curves and/or gaps between curves found"**. It shows practical methods to clean up and repair control geometry so that 3-axis toolpaths (especially Horizontal Roughing/Wrapping) can be generated successfully.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:24	Identifying the Error	Triggering the "Open curves and/or gaps" error during Horizontal Roughing
1:14	Failed Method: Extract Curve from Face Edge	Why directly extracting edges from faces often produces invalid geometry
2:20	Successful Method: Flat Area Command	Using the "Flat Area" command to generate clean boundary curves
3:29	Cleaning Up Geometry	Deleting unnecessary curves, exploding, and removing tiny invalid segments
5:15	Merging Curves	Combining cleaned curves into single continuous closed loops

Time	Topic / Section	Description
5:37	Verifying Closed Loops	How to check that a curve is properly closed (single endpoint indicator)
6:11	Toolpath Generation & Tool Selection	Fixing “invalid arguments” error by choosing the correct tool (Flat Mill vs Ball Mill)

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from this tutorial:

1. **Face Edge Extraction Often Fails** — Directly using “Extract Curve from Face Edge” on complex or imported geometry frequently produces broken, self-intersecting, or gapped curves that cause toolpath errors.
2. **Use “Flat Area” Command Instead** — The **Flat Area** command (in the Curve Modeling tab) is much more reliable for generating clean, accurate boundary curves around flat faces.
3. **Explode + Delete Tiny Segments** — After creating curves, explode them and look for small, invalid line segments (often shown as tiny squares). These are a common hidden cause of “open loop” errors — delete them.
4. **Always Merge Curves** — After cleaning, select all related curve segments and use the **Merge** command. A properly merged closed curve should show only **one endpoint marker**.
5. **Verify Before Generating Toolpaths** — A quick visual check (only one endpoint visible) can save you from repeated failed toolpath generations.
6. **Tool Choice Matters** — Sometimes the error is **not** geometry-related. In this case, switching from a **Ball Mill** to a **Flat Mill** resolved an “invalid arguments” error for Horizontal Roughing.
7. **Layer Management Helps** — Keep repaired control curves on a separate, clearly named layer so you can easily toggle visibility or reuse them later.

Key Takeaways

This short video provides a **reliable workflow** for fixing one of the most frustrating errors in MecSoft CAM:

“Open curves and/or gaps between curves found”

Recommended Repair Process:

- Use **Flat Area** command instead of direct edge extraction
- Explode curves and delete tiny invalid segments

- Merge all segments into single continuous curves
- Verify that closed curves have only one endpoint
- Ensure you're using the correct tool type for the operation

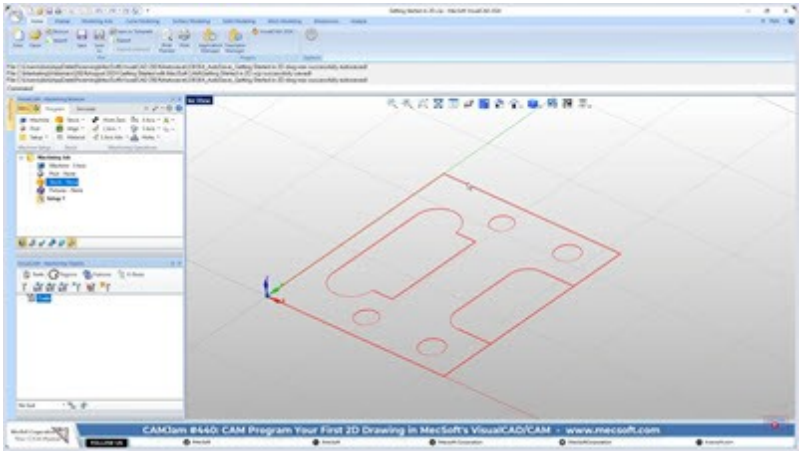
This method works especially well when dealing with imported geometry, complex molds, or parts with many flat faces.

5.8 CAM a 2D Drawing

Link: https://youtu.be/InX4c8GcjK4

This is an excellent **beginner-friendly tutorial** that walks through the complete workflow of programming a 2D drawing in **VisualCAD/CAM**. It clearly explains the challenges of working with pure 2D geometry (no automatic depth information) and shows how to properly set up stock, work zero, and cut levels.

Source Files: **Refer to the CAMJam 2025 Files Archive for Matching Source Files**



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:24	Introduction & Interface Overview	Why VisualCAD/CAM is ideal for 2D shop-floor work
0:50	Geometry Preparation	Merging multiple curve segments into single continuous curves
3:40	Stock & Work Zero Setup	Creating box stock using "Copy Model Bounding Box" and setting Work Zero at top Southwest corner

Time	Topic / Section	Description
5:00	2-Axis Pocketing Operation	Selecting the region and creating the first pocketing operation
5:30	Critical: Cut Level Configuration	Setting “Location of Cut Geometry = Pick Top” and defining total cut depth manually
7:30	Debugging & Troubleshooting	Fixing incorrect toolpath direction by adjusting geometry position and stock settings
9:00	Start Point Control	Changing the pocketing start location from inside to outside perimeter
10:45	2-Axis Facing Operation	Creating a facing pass to clear the top surface
11:45	Best Practices & Recommendations	Why 3D models are preferred and how to plan cut levels when using 2D drawings

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from this tutorial:

1. **Always Merge Curves First** — Before creating any toolpath, merge all separate curve segments into single continuous closed curves. This prevents selection errors and ensures clean toolpaths.
2. **Use “Pick Top” for 2D Drawings** — Since 2D drawings have no Z-depth information, always set **Location of Cut Geometry = Pick Top**. This lets you manually define the starting Z-height (usually the top of stock).
3. **Set Work Zero at Top Southwest Corner** — This is the most intuitive and reliable convention when working with 2D drawings. It makes all depth values positive and easy to understand.
4. **Manually Define Cut Depths Carefully** — With 2D geometry, you must manually calculate and enter the total cut depth. The presenter strongly recommends **sketching the part and cut levels on paper first** to avoid confusion.
5. **Move Geometry to Align with Stock Top** — If the toolpath is cutting in the wrong direction (e.g., upward instead of downward), move the geometry up so it sits at the top of the stock, then regenerate.
6. **Change Start Points When Needed** — You can easily change where the pocketing toolpath begins (inside vs. outside) by adjusting the start point settings.

7. **Use 3D Models Whenever Possible** — The presenter emphasizes that 3D models are far easier to work with because they automatically provide depth and surface information. Only use 2D drawings when a 3D model is not available.
8. **Simulate After Every Major Change** — Always run cut material simulation after adjusting stock, work zero, or cut levels to verify the toolpath behaves correctly.

Key Takeaways

This video is one of the **best beginner resources** for learning how to program **pure 2D drawings** in VisualCAD/CAM. It clearly explains the main challenge of 2D work (manual depth management) and provides a reliable workflow:

1. Merge curves
2. Set stock + Work Zero at top Southwest
3. Use “Pick Top” for cut levels
4. Carefully define depths
5. Simulate and adjust as needed

Best Practice Summary: When working with 2D drawings, always plan your cut levels on paper first. Use 3D models whenever possible for automatic depth and surface data.

Misc. Videos

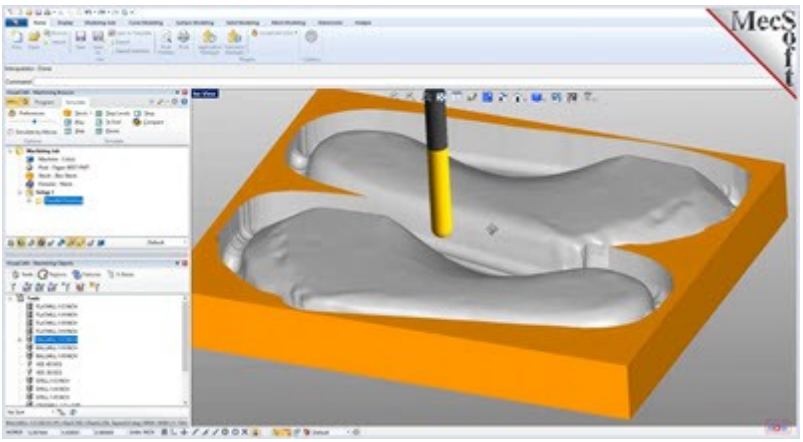
Enter topic text here.

6.1 Scanned Orthotics

Link: https://youtu.be/Vkj3qAcXbbU

This video demonstrates a complete, practical workflow for machining **scanned orthotics** (foot insoles) from STL mesh data using **VisualCAD/CAM**. It focuses heavily on handling mesh geometry, automation with Knowledge Bases, and efficient 3-axis toolpath strategies.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction	Overview of the orthotics machining workflow
0:46	Importing STL Files	Drag-and-drop import and file format support
2:06	Unit Verification & Scaling	Checking and correcting units (mm to inches)
3:50	Orienting the Part	Using Orient Part tool to flip the model
5:44	Mirroring for Left/Right Pairs	Creating mirrored copies and alternating orientation
6:58	Organizing with Layers	Using layers for part management and identification

Time	Topic / Section	Description
9:15	Machine, Post & Stock Setup	3-axis setup, post selection, and stock definition
11:50	Tool Definition	Setting up ball mills and feeds/speeds
14:50	Parallel Finishing Toolpath	Creating and simulating parallel finishing
22:00	Post-Processing & Documentation	Generating G-code and shop setup sheets
24:50	Knowledge Bases for Automation	Saving operations and using selection rules
28:00	Silhouette & 3D Offset Pocketing	Creating boundaries and using 3D offset pocketing
35:00	Working with Negative Casts	Mirroring, rotating, and nesting second parts

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the video:

1. **Always verify units immediately after importing STL files** — STL files contain no unit information, so check and scale the model right away.
2. **Use the "Orient Part" command** to quickly flip models so the bottom faces up for machining.
3. **Mirror parts and alternate orientation** (180° rotation) to improve nesting efficiency and save material.
4. **Organize parts on separate layers** — This makes it easy to manage multiple patient jobs and toggle visibility.
5. **Use Knowledge Bases + Selection Rules** for automation — Save your finishing operations once, then automatically apply them to new orthotics.
6. **Silhouette tool is excellent for mesh boundaries** — It creates clean curves around scanned parts for containment and pocketing.
7. **3D Offset Pocketing works very well on orthotics** — It provides consistent step-over and excellent surface finish on curved surfaces.
8. **Use the VisualCAM Mesh module for repair** — Tools like *Auto Fix*, *Cap*, and *Reduce* are essential for cleaning up scanned STL data.

9. **Align stock to the top of the part** when machining from one side to preserve the original shape.
10. **Generate shop documentation** (HTML or Excel) after posting — Very useful for operators on the shop floor.
11. **Save recurring operations as Knowledge Bases** — This dramatically reduces programming time for similar orthotics jobs.

Key Takeaways

This video shows a **complete, repeatable workflow** for machining custom orthotics from 3D scans using **VisualCAD/CAM**. The combination of **mesh handling**, **Knowledge Base automation**, and **3D Offset Pocketing** makes it possible to efficiently produce high-quality custom insoles with minimal manual intervention.

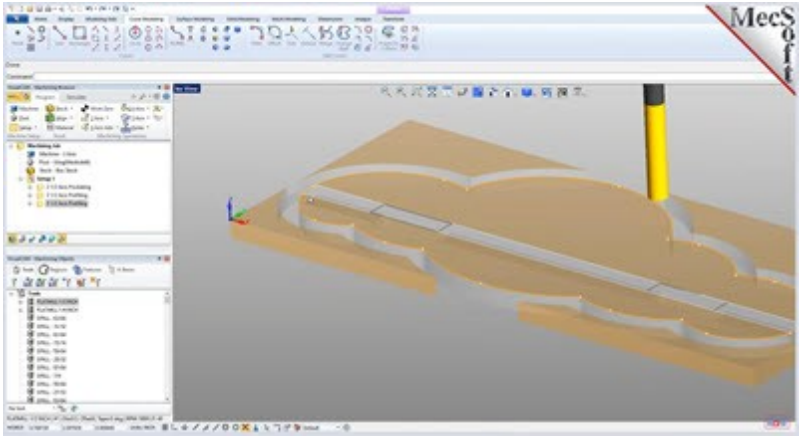
Best suited for: Orthotics labs, podiatry clinics, and manufacturers working with scanned foot data.

6.2 2.5 Axis Pocketing & Profiling

Link: https://youtu.be/-o8fQ2uQypA

This is a practical beginner-to-intermediate tutorial focused on the two most commonly used 2.5-axis operations in VisualCAD/CAM: Pocketing and Profiling. It walks through the full workflow from geometry selection to finished toolpaths with clear explanations of key settings.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction	Overview of the tutorial and what will be covered
0:45	Geometry Requirements	Importance of merged, closed curves for pocketing
1:50	Creating a Pocketing Operation	Selecting the region and setting basic parameters
4:20	Cut Levels & Depth Control	Using “Pick Top”, Total Cut Depth, and Step Down
7:10	Entry/Exit Methods	Ramp entry vs Helical entry and when to use each
9:40	Step Over & Stock to Leave	Recommended values and how they affect finish
11:30	Creating a Profiling Operation	Selecting curves and setting cut side
13:20	Cutter Compensation (G41/G42)	When and how to use it correctly
15:10	Simulation & Verification	Running cut material simulation and checking results
16:45	Common Mistakes & Best Practices	Tips for avoiding errors in 2-axis operations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the video:

1. Always merge curves before creating toolpaths — Un merged curves are one of the most common causes of failed or broken toolpaths in pocketing and profiling.
2. Use “Pick Top” for 2D geometry — When working with 2D drawings, always set Location of Cut Geometry = Pick Top so you can manually control the starting depth.
3. Ramp Entry is usually safer than Helical — Use ramp entry for most general pocketing. Reserve helical entry for deep pockets or when you want the cleanest possible entry.

4. Leave stock on the walls during roughing — A small positive stock to leave (e.g., 0.010"–0.020") on profiling operations protects the final surface and allows for a cleaner finishing pass.
5. Use Cutter Compensation only on the final pass — Enable G41/G42 only on finishing operations, and always ensure you have a linear entry move for the compensation to engage properly.
6. Step Over should be 40–60% of tool diameter for roughing and 10–25% for finishing (depending on material and desired surface quality).
7. Simulate with different colors — Use different colors for roughing and finishing operations in simulation to easily see what each pass is removing.
8. Check cut side carefully — In profiling, always verify whether the tool is cutting inside or outside the curve before generating the toolpath.
9. Start with conservative depths — When learning, use smaller step-down values. It's better to take more passes than to risk tool breakage or poor finish.

Key Takeaways

This video is an excellent, no-nonsense guide to the two core 2.5-axis operations in VisualCAD/CAM. It emphasizes clean geometry preparation and proper depth control — two of the most important fundamentals for reliable 2-axis machining.

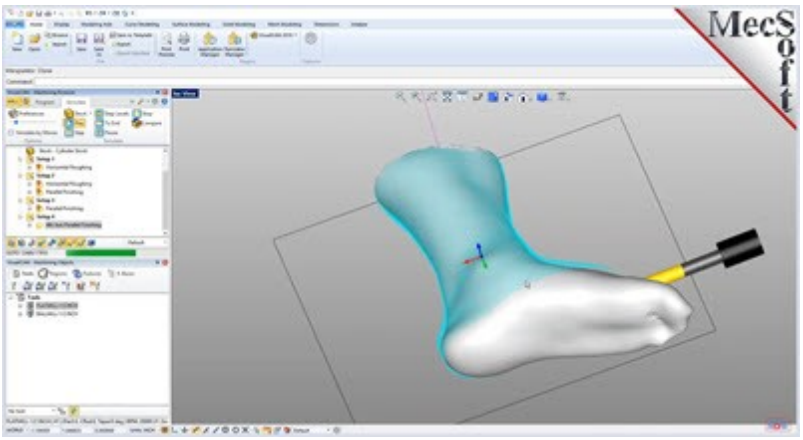
Best suited for: Beginners and intermediate users who want to master pocketing and profiling with confidence.

6.3 4 Axis Scan Data

Link: https://youtu.be/g3KShOm3xuU

This CAMJam episode demonstrates a complete workflow for machining a **scanned part (STL mesh)** using **4-axis machining** in VisualCAD/CAM. It covers importing and preparing scan data, setting up a 4-axis machine, roughing, indexed finishing, and continuous 4-axis finishing, plus automation with Knowledge Bases.

Source Files: **Refer to the CAMJam 2025 Files Archive for Matching Source Files**



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction	Overview of using 4-axis with scanned STL data
0:46	Importing the STL File	Drag-and-drop import and initial checks
2:06	Unit Verification & Scaling	Checking units and correcting scale
3:50	Orienting the Part for 4-Axis	Rotating the model to align with the rotary axis
5:44	4-Axis Machine Setup	Defining machine type, rotary axis, and post processor
9:15	Stock Definition	Creating and aligning cylindrical stock
11:50	Roughing Operation (3-Axis)	Linear pattern roughing with containment
14:50	Indexed 3-Axis Finishing	Parallel finishing on multiple sides
22:00	Reorienting for Continuous 4-Axis	Aligning the part to the rotation center
28:00	Continuous 4-Axis Finishing	Parallel finishing with helical cut pattern
35:00	Simulation & Verification	Checking for collisions and tool holder interference

Time	Topic / Section	Description
40:00	Post-Processing	Generating G-code
44:00	Saving to Knowledge Base	Automating the workflow for future similar parts

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the video:

1. **Always check and correct units immediately** after importing an STL file — scanned data often comes in the wrong units.
2. **Use the graphical manipulator** to quickly rotate and position the part so it aligns properly with the 4th axis.
3. **For continuous 4-axis**, the part **must** be centered on the rotation axis. Reorient the model if needed.
4. **Rough with 3-axis strategies first**, then switch to indexed or continuous 4-axis for finishing. This is often more efficient and stable.
5. **Use cylindrical stock** aligned to the part to minimize material waste and improve Fixturing.
6. **Simulate with tool holder shown** — Watch for collisions between the tool holder and the part/stock, especially during 4-axis rotations.
7. **Helical cut pattern** is recommended for continuous 4-axis finishing as it produces smoother motion and better surface finish.
8. **Save your workflow as a Knowledge Base** — This is extremely useful when machining multiple similar scanned parts (e.g., medical or custom components).
9. **Mesh repair tools** (in the Mesh module) are often needed before machining scanned data — use Auto Fix, Cap, and Reduce when necessary.
10. **Test your post-processor** thoroughly with 4-axis output before running on the machine.

Key Takeaways

This CAMJam episode provides a **clear, real-world workflow** for machining scanned parts using 4-axis. It shows that even complex mesh data from scanners can be efficiently machined by combining 3-axis roughing with indexed and continuous 4-axis finishing.

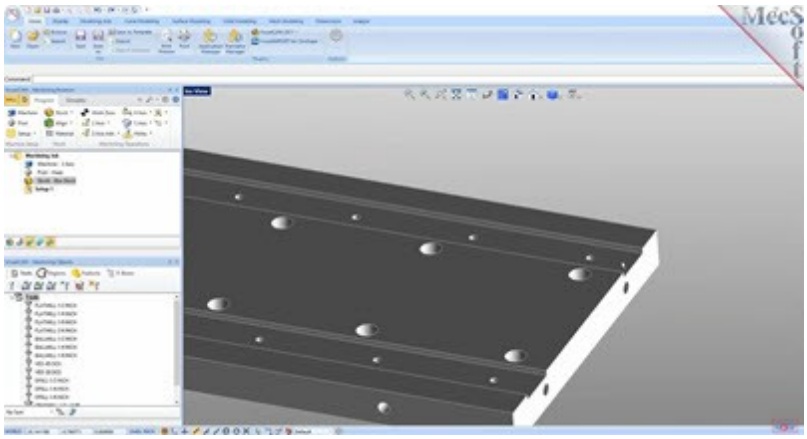
Best suited for: Users working with scanned data (medical, dental, custom parts, reverse engineering) who want to take advantage of 4-axis capabilities.

6.4 2.5 Axis Standard

Link: https://youtu.be/1P_u-08lMrU

This CAMJam episode provides a detailed, hands-on walkthrough of 2.5-axis machining in VisualCAD/CAM. It covers the full workflow from geometry preparation to finished toolpaths, including pocketing, profiling, facing, drilling, and basic finishing strategies.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of 2.5-axis machining workflow
2:30	Geometry Preparation	Importing, merging curves, and creating regions
8:00	Stock & Work Zero Setup	Defining stock and setting machining origin
12:00	Facing Operation	Top facing with ramp entry and cleanup pass
18:30	Pocketing Operation	Step-over, stock to leave, ramp/helical entry, and cleanup pass
28:00	Profiling Operation	Cut side selection, cutter compensation, and entry/exit control
36:00	Drilling Operations	Spot drill, peck drill, and tapping cycles
44:00	Basic Finishing	Parallel finishing and when to use it

Time	Topic / Section	Description
50:00	Simulation & Verification	Cut material simulation and checking results
55:00	Post-Processing	Generating G-code and common post settings
1:00:00	Common Mistakes & Best Practices	Tips for reliable 2.5-axis programming

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the video:

1. **Always merge curves before machining** — Un merged curves are one of the top causes of failed or broken toolpaths.
2. **Use "Pick Top" when working with 2D drawings** — Since 2D geometry has no automatic Z information, always set Location of Cut Geometry = Pick Top.
3. **Ramp or Helical entry is strongly preferred** — Avoid plunging directly into material unless absolutely necessary.
4. **Leave stock on walls for finishing** — A small positive stock-to-leave (0.010"–0.020") on profiling operations allows for a cleaner final pass.
5. **Enable Cutter Compensation only on the final pass** — G41/G42 works best on finishing operations with a linear entry move.
6. **Use Predefined Regions** — Create and name regions for common feature groups (holes, pockets, profiles) to speed up future programming.
7. **Simulate after every major operation** — Especially after roughing and before drilling or finishing.
8. **Check cut side carefully in profiling** — Always verify whether the tool is cutting inside or outside the curve.
9. **Start conservative with depths and step-overs** — It's better to take more passes than risk tool damage or poor finish.
10. **Use the Cleanup Pass option in pocketing** — This removes leftover stock in corners and improves final finish.
11. **Group similar operations** — Organize your Machining Job tree by operation type or by tool for easier management.

Key Takeaways

This video is one of the best foundational tutorials for 2.5-axis machining in VisualCAD/CAM. It emphasizes clean geometry preparation, proper depth control, and consistent simulation — three fundamentals that prevent most common 2.5-axis problems.

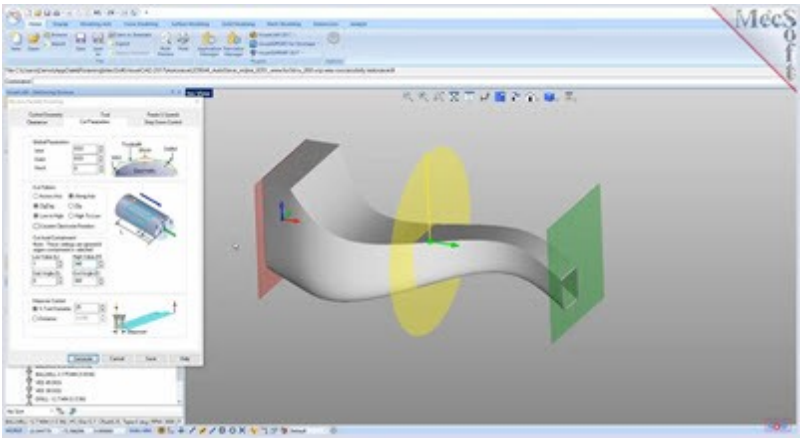
Best suited for: New and intermediate users who want to build a solid understanding of pocketing, profiling, facing, and drilling.

6.5 4 Axis Furniture

Link: https://youtu.be/PiyvjZ5gnF8

This video demonstrates a complete 4-axis machining workflow using a furniture design component (likely a chair leg or decorative wooden part) in VisualCAD/CAM 2017. It shows both indexed 4-axis and continuous 4-axis strategies applied to a real-world design project.

Source Files: **Refer to the CAMJam 2025 Files Archive for Matching Source Files**



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Project Overview	Overview of the furniture part and 4-axis goals
2:30	Part Analysis & Setup	Reviewing the 3D model and planning machining strategy
6:45	4-Axis Machine Definition	Setting up rotary table, axis orientation, and post processor
11:20	Stock Definition	Creating cylindrical stock aligned to the part

Time	Topic / Section	Description
14:50	3-Axis Roughing	Horizontal roughing before switching to 4-axis
20:10	Indexed 4-Axis Finishing	Using Rotate Table Setup at multiple angles (0°, 90°, 180°)
28:30	Continuous 4-Axis Finishing	4-Axis Parallel Finishing with wrap method
35:40	Toolpath Simulation	Full simulation with rotary axis movement
40:20	Post-Processing	Generating 4-axis G-code
44:00	Design Considerations for 4-Axis	Tips for designing furniture parts for rotary machining
48:00	Q&A and Summary	Final thoughts and recommendations

Key Tips & Best Practices Extracted

Here are the most valuable practical takeaways from the video:

1. **Center the part on the rotary axis** — For continuous 4-axis machining, the part must be properly aligned with the rotation center to avoid errors and collisions.
2. **Rough with 3-axis first, then finish with 4-axis** — This hybrid approach is often more efficient and stable than trying to do everything with 4-axis.
3. **Use cylindrical stock** — Aligning stock to the part's center reduces material waste and improves Fixturing.
4. **Indexed 4-axis is simpler and more reliable for many furniture components** — Use "Rotate Table Setup" to machine multiple sides without complex continuous motion.
5. **Continuous 4-axis (Wrap method)** works best on parts with consistent radial features.
6. **Simulate with full machine motion** — Always check for tool holder collisions during rotary movements.
7. **Design with 4-axis in mind** — Parts with rotational symmetry or features around a central axis are ideal for 4-axis machining.
8. **Use Knowledge Bases** to save your 4-axis setups and operations for similar furniture projects.
9. **Test your post-processor thoroughly** with 4-axis output before running production parts.

Key Takeaways

This video shows a real-world application of 4-axis machining on a furniture component. It highlights the practical benefits of combining 3-axis roughing with both indexed and continuous 4-axis finishing to efficiently machine complex wooden parts.

Best suited for: Furniture designers, woodworkers, and manufacturers interested in using 4-axis CNC for custom or production furniture components.

Product Features

7.1 What's New in 2026

Link: https://www.youtube.com/watch?v=u2TqbSTF6c4

This webinar provides a detailed walkthrough of the major new features and workflow improvements introduced in RhinoCAM 2026 and VisualCAD/CAM 2026. The presentation focuses on automation, advanced 4-axis machining strategies, simulation performance, Deburring enhancements, post-processing updates, and overall productivity improvements. Throughout the session, the presenters also share practical machining tips and best practices for integrating these new tools into real-world CNC workflows.

Source Files: 20.26.02 Open Webinar What's New in MecSoft CAM 2026.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction and Agenda Overview	Release overview and what's new in 2026
4:26	Five-Axis 3+2 Auto Roughing	Automatic indexed roughing setups based on stock/part analysis
7:21	4-Axis Extrusion Machining	Profile + Rail method for constant cross-section parts
12:38	4-Axis Extrusion Projection Machining	Extrude profile along rail then project onto 3D model

Time	Topic / Section	Description
16:37	2.5-Axis Automatic Deburring / Chamfering	Edge detection + automatic toolpath generation (works with ball mills)
20:28	GPU-Accelerated Tri-Dexel Simulation Model	5–10× faster cut material simulation (NVIDIA GPUs)
31:01	Fixture Support for Individual Setups	Assign different fixtures/clearances per setup
32:22	Arc Feed Rate Optimization	Reduce feed before/after arcs for better finish & tool life
35:19	Drill Point Sorting Between Operations	Start next operation at closest point to previous cut
37:32	Miscellaneous Enhancements	Flats-only output, fixed tool axis, angle limit handling
41:03	Post-Processor Enhancements	Python posts for drill banks, dual heads, non-standard formats
42:34	Expanded API Support	New functions for setups, operations, simulation, and Knowledge Bases
44:32	Live Demos (Main Features)	3+2 Auto Roughing, 4-Axis Extrusion/Projection, Deburring, etc.
1:10:36	Live Demos (Post-Processors)	5-axis G68.2, drill banks, dual heads, Holzher, WoodFlash, Flow Master
1:22:22	Conclusion & Release Timeline	Mid-March 2026 release

Key Tips & Best Practices Extracted

Here are the most valuable new capabilities and practical takeaways from the 2026 release:

- **3+2 Auto Roughing is a Game-Changer** — The new operation automatically analyzes the stock and part, determines the optimal indexing angles, creates multiple setups, and generates adaptive roughing toolpaths — all with almost zero manual input. Ideal for complex prismatic or semi-prismatic parts.

- **4-Axis Extrusion Methods Open New Possibilities** —
    - Extrusion Machining (Profile + Rail) is perfect for parts with constant cross-section (no need for a full 3D model).
    - Extrusion Projection Machining solves the long-standing limitation of features that extend above or below the rotary axis by projecting the extruded profile onto the actual 3D geometry.
- **2.5-Axis Auto Deburring Works with Ball Mills** — You no longer need special V-mills or chamfer tools. The new operation automatically detects edges and generates safe deburring/chamfering paths with full gouge checking.
- **Tri-Dexel Simulation is a Major Performance Win** — If you have an NVIDIA GPU (especially in Professional+ configurations), enable the new Tri-Dexel simulation model. It delivers 5–10× faster simulation with excellent accuracy — especially noticeable on complex multi-setup jobs.
- **Per-Setup Fixtures & Clearances** — You can now assign different fixtures and clearance distances to individual setups instead of applying one setting to the entire job. This is extremely useful for tombstone or multi-part fixtures.
- **Arc Feed Rate Optimization Improves Finish & Tool Life** — Reduce feed rates automatically before and after arcs (corners). This is configurable by arc length and radius and helps prevent chatter and extend tool life on high-speed finishing passes.
- **Drill Point Sorting Reduces Non-Cutting Time** — The new "Drill Point Sorting Between Operations" option makes the next operation start at the closest point to where the previous operation finished — a simple but effective way to shorten cycle time on parts with many holes or features.
- **Python Post-Processors Are Now Extremely Powerful** — 2026 significantly expands Python post support, including full drill bank control, dual-head machines, and non-standard G-code formats (Holzher, WoodFlash, Flow Master waterjet, etc.). Custom post development is now much more accessible.
- **Expanded API for Automation** — New API functions allow deeper integration with external scripts for setups, operations, simulation, and Knowledge Base workflows — great for shops building custom automation pipelines.

Key Takeaways

MecSoft CAM 2026 focuses heavily on automation, performance, and expanding 4-axis/5-axis capabilities while making daily workflows faster and smarter.

Biggest highlights of the 2026 release:

- This is one of the most significant releases in recent years, especially for users doing multi-sided work or running long simulation jobs.

- Use automated 3+2 roughing setups to reduce programming time.
- Apply extrusion machining for faster and more flexible rotary machining workflows.
- Leverage GPU acceleration to dramatically improve simulation performance.
- Automate deburring operations to reduce manual edge selection.
- Upgrade legacy posts using expanded Python post-processing capabilities.
- Use expanded APIs to automate repetitive CAM workflows.

7.2 What's New in 2025

Link: https://www.youtube.com/watch?v=TZqTqwVcdgY

This webinar introduces the major new features and enhancements included in MecSoft CAM 2025 for RhinoCAM and VisualCAD/CAM. The presentation covers improvements across 2½ Axis, 3 Axis, 4 Axis, and 5 Axis machining, along with updates to simulation, post processing, usability, automation, and SDK functionality. Throughout the webinar, the presenters also share practical workflow tips and recommendations for improving machining efficiency, tool life, surface finish, and overall programming productivity.

Source Files: 20.25.02 Open Webinar Whats New in Version 2025.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2025 release
0:40	2½-Axis Milling Enhancements	Cutter Compensation in Hole Pocketing + Thread Milling improvements

Time	Topic / Section	Description
2:10	3-Axis Milling Enhancements	Feedrate Optimization, Faceting Controls, Drive Curves + Surfaces, Corner Cleanup Loops, Clear Flats with Stock Allowance, Adaptive Roughing improvements
4:45	5-Axis Milling Enhancements	Machining on either side in Curve Projection (offset capability)
5:20	Hole Making Enhancements	Rigid Tapping (G84) support
5:50	Simulation Enhancements	Machine Tool Linear Limits detection and error flagging
6:20	Usability Enhancements	Copy/Clone naming, separate simulation tolerance, reset preferences, maintain control geometry order
7:10	Post-Processing & SDK Enhancements	Python 3 support, asynchronous writing, new variables
8:00	Conclusion & Release Info	Final summary and availability

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the 2025 release:

1. **Cutter Compensation Now Works in Hole Pocketing** — You can now use G41/G42 in hole pocketing and profiling operations for tighter control and better finish on production holes.
2. **Thread Milling Got Smarter** — New controls for **Start Depth (Relief Depth)**, pitch (per mm or per inch), and **Number of Starts** give you much more precise control over thread creation.
3. **Feedrate Optimization is Now Per-Operation** — Corner feed reduction settings have moved directly into the operation dialog (instead of global preferences), making it easier to fine-tune per toolpath.
4. **Faceting Controls Improve Toolpath Quality** — New mesh aspect ratio and minimum edge length settings in 3-axis operations help prevent jagged or erratic toolpaths on complex ruled surfaces.

5. **Drive Curves + Drive Surfaces Can Now Be Used Together** — In operations like Parallel Finishing, you can select both surfaces and curves as drive geometry — very useful for machining between nested regions.
6. **Corner Cleanup Loops in Projection Pocketing** — New option (Professional configuration) adds automatic cleanup loops at sharp corners to reduce material retention and machine jerking.
7. **Clear Flats Now Supports Stock Allowance** — You can now leave a specified stock allowance on flat areas when using the "Clear Flats" option, allowing for a dedicated finishing pass.
8. **Adaptive Roughing Improvements** — Helical entries at every cut level and arc motions for transfers/exits make Adaptive Roughing more efficient and tool-friendly.
9. **Rigid Tapping (G84) is Now Supported** — Higher-accuracy tapping with synchronized spindle and feed control.
10. **Machine Tool Linear Limits Detection** — Define X/Y/Z travel limits in the Machine Tool setup. The software now visually shows the limits and flags any violations as errors during simulation.
11. **Copy/Clone Naming is Now Customizable** — You can disable or change the automatic "-copy" and numbering behavior when cloning operations.
12. **Maintain Order of Control Geometry** — New option prevents the software from automatically merging or reordering selected curves/regions — useful when order matters.
13. **Python 3 SDK Support** — Major upgrade from Python 2, plus non-interactive execution for "lights-out" automation.

Key Takeaways

MecSoft CAM 2025 focuses on **practical daily productivity** rather than flashy new modules. The biggest wins are:

- This is a solid, refinement-focused release that makes existing workflows faster, safer, and more precise.
- More control and intelligence in 3-axis finishing (faceting, corner cleanup, drive geometry flexibility)
- Better automation and safety (rigid tapping, machine limits detection, feed optimization)
- Improved usability (naming, ordering, simulation tolerance)
- Use cutter compensation in hole pocketing for improved dimensional control.
- Apply feedrate optimization to improve surface finish and extend tool life.
- Use updated faceting controls to reduce waviness in 3-axis machining.

- Validate rotary and linear machine limits before posting code.
- Take advantage of rigid tapping and updated hole-making workflows.
- Use simulation enhancements to detect machine violations early.
- Leverage Python 3 SDK support for automation and customization.
- Apply usability improvements to streamline daily CAM workflows.

7.3 What's New in 2024

Link: https://www.youtube.com/watch?v=-J3bJb29vwM

This webinar highlights the new features and enhancements in MecSoft CAM 2024 for VisualCAD/CAM and RhinoCAM. Presented by the MecSoft team, it walks through usability improvements, new machining capabilities (including 5-axis drilling and laser tools), high-speed options, mold-making enhancements, 4/5-axis flexibility, and licensing updates. The session includes explanations and demonstrations of how these updates improve workflow efficiency, precision, and user experience.

Source Files: 20.24.02 Open Webinar Whats New in MecSoft CAM 2024.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2024 release

Time	Topic / Section	Description
3:30	Usability Enhancements	Posting order by tool number, Save/Load Post with file
5:45	5-Axis Enhancements	True 5-Axis Drilling (tool normal to surface)
6:15	3-Axis Enhancements	Skim Clearance with In process Stock, Negative Offsets in Face Off, High-Speed Machining in Facing, Exact Z-Level Distances
8:55	Tool Contact Condition (Mold Making)	New “Contact” surface condition to stop tool exactly at parting line
15:55	4-Axis & 5-Axis Enhancements	4-Axis rotation on Z-axis, Recalculate Angles after reordering
18:15	Licensing Enhancements	Improved Network Lock License Manager
19:15	Live Demo: Posting Order & Save/Load Post	Tool-number-based posting and locking posts to files
23:43	Live Demo: 5-Axis Drilling	True 5-axis drilling on curved surfaces
25:40	Live Demo: 2-Axis Laser Tools & Negative Offsets	New Laser Tool definition and tightening toolpaths
28:30	Live Demo: High-Speed Facing	Smooth arc entries/exits in facing operations
30:30	Live Demo: Tool Contact Condition	Preventing flash on parting lines in mold cavities
39:10	Live Demo: Exact Z-Level & Skim Clearance	Precise step-downs and smarter retracts
41:50	Live Demo: 4-Axis Z-Axis Rotation	Setting 4th axis to rotate around Z (C-axis)
44:15	Q&A and Licensing Details	Network Lock setup, Cloud vs Network licenses, Rhino 8 compatibility

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the 2024 release:

- **Post by Tool Number (Ascending/Descending)** — Enable this in CAM Preferences > Post-Processing to automatically reorder operations by tool number when posting. This dramatically reduces tool changes on the machine without manually reorganizing your Machining Job tree.
- **Always Save & Load Post with the File** — Check both "Always Save Post to File" and "Always Load Post from File" in CAM Preferences. This locks the correct post-processor to each job file, preventing errors when moving files between computers or after software updates.
- **Tool Contact Condition is a Major Win for Mold Makers** — The new "Contact" surface condition makes the tool stop exactly when the ball touches the perimeter edge (parting line). This prevents the tool from riding up over the parting line (which causes flash) and often eliminates the need for masking surfaces.
- **Skim Clearance Now Considers In-Process Stock** — Retracts are now calculated based on remaining stock, allowing much lower (and faster) retract heights while still avoiding collisions.
- **Negative Offsets in Face Off** — You can now enter negative values for cut area extension. This tightens the toolpath around the part edge, saving significant machining time on facing operations.
- **High-Speed Arc Entries/Exits Now Work in Facing** — Facing operations now benefit from the same smooth 2D/3D arc entries, exits, and transfers previously available only in pocketing — reducing machine jerk and improving finish.
- **Exact Z-Level Distances** — You can now specify precise cut distances for Z-level operations instead of being forced to cut at the absolute bottom level. This gives much more control over step-downs.
- **True 5-Axis Drilling** — The tool can now stay normal to the surface during drilling on curved or angled geometry (not just indexed 5-axis drilling).
- **4-Axis Rotation on Z-Axis** — You can now assign the 4th axis to rotate around the positive or negative Z-axis (C-axis style), expanding 4-axis capabilities for certain machine configurations.
- **Recalculate Angles After Reordering** — After moving operations within a setup, use the new Recalculate Angles option to update machine tool kinematics without regenerating the entire simulation.

- **Improved Network Lock License Manager** — The new interface automatically creates firewall rules and generates registry files for client machines, making network license setup much easier.

Key Takeaways

MecSoft CAM 2024 is a strong refinement release focused on usability, mold-making productivity, and workflow reliability.

Biggest highlights:

- Tool-number-based posting + locked post-processors (huge time saver)
- Tool Contact Condition — game-changing for mold quality
- Skim Clearance with In-Process Stock + Negative Offsets (faster machining)
- True 5-Axis Drilling and expanded 4-Axis flexibility

This release makes daily work faster, safer, and more consistent — especially for users doing mold work or running production jobs with many tool changes.

7.4 What's New in 2023

Link: https://www.youtube.com/watch?v=__TbLLJhLI0

This is the official **"What's New"** webinar for the **2023 release** of RhinoCAM and VisualCAD/CAM. It covers usability improvements, new CAD modeling tools in VisualCAD, major enhancements across 2.5-axis, 3-axis, 4-axis, 5-axis, and Turning modules, plus post-processing and licensing updates.

Source Files: 20.23.01 Open Webinar Whats New in MecSoft CAM 2023.zip

Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2023 release focus areas
3:00	VisualCAD/CAM Curve Modeling Enhancements	New Rectangle (3 methods), Ellipse, Closed Curve, Split commands + display improvements
7:00	Usability Enhancements	Re sizable dialogs, Save/Load Post with file, Toolpath Editor edits marking, Stock Z-offset
9:30	2.5-Axis Enhancements	Drilling along curves, Profiling Clustering, Gouge Checking, Saw Cutting improvements
13:00	3-Axis Enhancements	End conditions on curve regions, Projected Spiral Offset Pocketing, Improved Horizontal Roughing
16:00	4-Axis & 5-Axis Enhancements	4-axis drilling along curves, 5-axis alternate rotation angles, Z-axis reorientation handling
19:00	Turning Module Enhancements	Geometry offset from X-axis, Linear retracts for ID machining
21:00	Post-Processing Enhancements	Legacy post macros, new variables, Programmable post improvements
23:00	Licensing Enhancements	New Network Log licensing, License Service Manager, diagnostics tools
25:00	Live Demonstrations	Full walkthrough of new features in VisualCAD/CAM 2023
56:00	Post-Processing & Licensing Deep Dive	Detailed explanation of changes and benefits

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the 2023 release:

1. **Drilling Along Curves is a Huge Time Saver** — You can now select any 2D or 3D curve (including surface edges and circles) and the software will automatically generate drill points along it. No more manually creating point arrays.

2. **Clustering in Profiling Prevents Tool Interference** — When profiling nested parts, the new **Clustering** option alternates inside/outside cutting to keep the tool from colliding with adjacent parts.
3. **Gouge Checking for Profiling** — A powerful new option that automatically calculates the safe outer perimeter of selected curves so the tool never gouges into neighboring geometry.
4. **Projected Spiral Offset Pocketing (Professional+)** — New cut pattern that combines spiral entry with offset pocketing for superior finish on complex pockets.
5. **Improved Horizontal Roughing ("Clear Flats")** — The software now ensures safe entry motions outside of material when clearing flat areas, preventing plunging into uncut stock.
6. **4-Axis Drilling Along Curves** — You can now drill along curves on 4-axis parts without manually creating indexed setups — the software handles the rotary motion automatically.
7. **5-Axis Alternate Rotation Angles** — The post can now output both the primary and alternate (180° offset) solutions for rotary axes, giving you more flexibility on machines with limited rotation range.
8. **Turning: Geometry No Longer Needs to Touch X-Axis** — You can now machine parts that are offset from the X-axis (great for parting-off operations behind the part).
9. **Linear Retracts for ID Machining** — ID turning now uses linear retracts to the clearance plane instead of diagonal moves, reducing the chance of tool interference.
10. **Save Post with File** — Enable **"Always Save Post to File"** and **"Always Load Post from File"** in preferences. This locks the correct post-processor to each job file and prevents errors when moving files between computers.
11. **New License Service Manager** — The 2023 licensing system includes a much-improved Network Log License Manager with better diagnostics and automatic firewall rule creation.

Key Takeaways

The **2023 release** focused heavily on **robustness and daily productivity** based on customer feedback. The biggest wins are:

- Significantly improved curve modeling and usability in VisualCAD
- Powerful new automation in 2.5-axis (drilling along curves, clustering, gouge checking)
- Better control and safety in 3-axis and 4/5-axis operations
- Major improvements to post-processing flexibility and licensing management

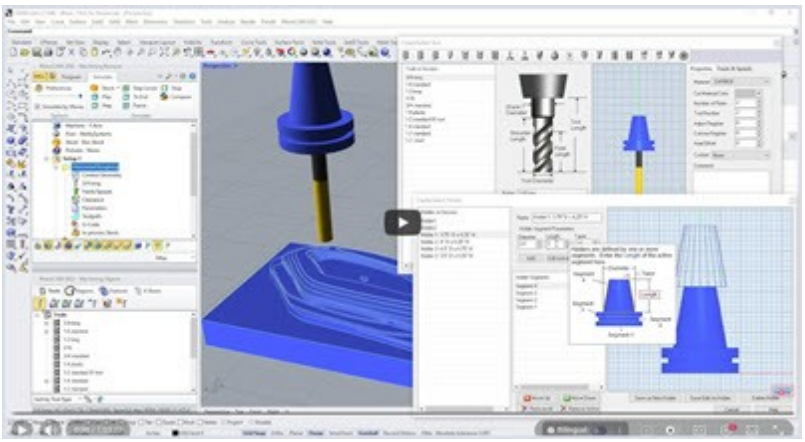
This was a very strong, refinement-focused release that made existing workflows faster and more reliable.

7.5 What's New in 2022

Link: https://www.youtube.com/watch?v=Qct4iFskXAc

This is the official “What’s New” webinar for the 2022 release of RhinoCAM and VisualCAD/CAM. It covers major usability improvements, new operations (including Thread Milling and Saw Cutting), significant post-processor advancements (Programmable Posts), the integration of the G-Code Editor module, Knowledge Base enhancements, and licensing changes.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2022 release
3:30	Usability Enhancements	Re sizable dialogs, improved icons, better workflow
7:00	Tooling Changes	New tool types and improved tool library management
11:00	Hole Making Enhancements	New Thread Milling operation and Hole Pocketing improvements
18:00	Milling Enhancements	New Saw Cutting operation and other 2.5/3-axis improvements

Time	Topic / Section	Description
24:00	Turning Module Enhancements	New features and improvements in the TURN module
29:00	Knowledge Base Enhancements	Better saving/loading and new automation capabilities
35:00	Post-Processor Changes	Programmable Post Processors (Python-based) + new macros
42:00	G-Code Editor Integration (AMS Exclusive)	Full integration into Milling module with real-time back-plotting and simulation
50:00	Licensing Changes	New licensing system and improvements
56:00	Live Demos & Q&A	Full walkthrough of major new features
1:01:48	Conclusion & Summary	Final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the 2022 release:

1. **G-Code Editor is Now Fully Integrated (AMS Subscribers)** — The G-Code Editor is no longer a separate module. It is built directly into the Milling module. You can now automatically back-plot and simulate the actual posted G-code in real time on your part — a massive improvement for catching post-processor errors before running on the machine.
2. **Programmable Post Processors (Python)** — 2022 introduced full Python-based programmable posts. This gives advanced users and post developers far more flexibility to create custom output for complex machines (drill banks, dual heads, special formats, etc.).
3. **New Thread Milling Operation** — A dedicated Thread Milling operation was added, giving much better control over thread creation compared to previous workarounds.
4. **New Saw Cutting Operation** — A dedicated operation for saw blades (disk cutting) with proper tool definition and containment options — very useful for panel saws and certain router applications.
5. **Hole Pocketing Improvements** — Better handling of large holes with improved helical entry and cleanup pass options.

6. **Save/Load Post with File** — A major usability win: You can now save the post-processor settings with the part file so the correct post is automatically loaded when the file is opened on any computer.
7. **Knowledge Base Enhancements** — Improved saving and loading of Knowledge Bases with better support for complex multi-operation workflows.
8. **Re sizable Dialogs** — Almost all dialogs became re sizable, significantly improving usability on high-resolution or multi-monitor setups.

Key Takeaways

The 2022 release was a major usability and power-user focused update. The two biggest highlights were:

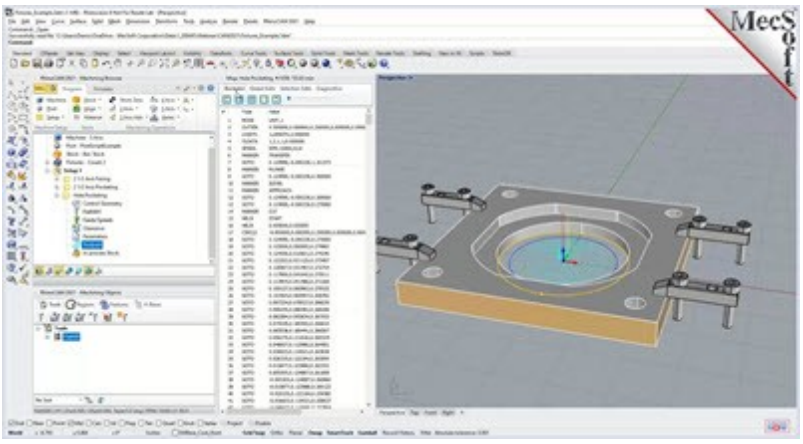
- This was one of the most well-received releases in recent years because it directly addressed long-standing user requests for better verification and customization.
- Full integration of the G-Code Editor into the Milling module (a game-changer for verifying actual posted code).
- Introduction of Programmable Post Processors (Python), giving users and developers unprecedented control over G-code output.
- Other improvements (Thread Milling, Saw Cutting, Save Post with File, re sizable dialogs) made daily work noticeably smoother and more reliable.

7.6 What's New in 2021

Link: https://youtu.be/jvVltCGiPJc

This is the official **"What's New"** webinar for the **2021 release** of RhinoCAM and VisualCAD/CAM. It covers major usability improvements, new operations, 4-axis/5-axis enhancements, post-processing updates, and performance gains.

Source Files: 20.21.01 Open Webinar Whats New in MecSoft CAM 2021.zip



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2021 release focus
3:20	Major Usability Improvements	Re sizable dialogs, new icons, better workflow
7:45	New 2.5-Axis Operations	Saw Cutting operation and other 2.5-axis enhancements
12:30	3-Axis Enhancements	Improved Horizontal Roughing, new finishing options
18:00	4-Axis & 5-Axis Enhancements	New 4-axis options and 5-axis improvements
24:00	Post-Processor Updates	New macros and improved post-processing
29:00	Knowledge Base Enhancements	Better saving and loading of Knowledge Bases
34:00	Performance & Simulation Improvements	Faster toolpath calculation and simulation
40:00	Licensing Changes	Updates to licensing system
45:00	Live Demonstrations	Full walkthrough of major new features
58:00	Q&A and Conclusion	Final questions and summary

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the **2021 release**:

1. **Re sizable Dialogs** — Almost every dialog in the software became re sizable. This was one of the most requested usability improvements and significantly improved the experience on high-resolution monitors.
2. **New Saw Cutting Operation** — A dedicated operation was added for saw blades (disk cutting). This made it much easier to program panel saws and certain router applications with proper tool definition and containment.

3. **Improved Horizontal Roughing** — Better handling of flat areas and improved entry/exit motions, especially when using the “Clear Flats” option.
4. **Knowledge Base Improvements** — Enhanced saving and loading of Knowledge Bases, making it easier to create and reuse automated workflows.
5. **Post-Processor Enhancements** — New macros were added, giving users more control over G-code output, especially for multi-axis and custom machines.
6. **Performance Optimizations** — Toolpath calculation and simulation were noticeably faster, especially on complex parts with many operations.
7. **4-Axis & 5-Axis Improvements** — Several refinements to indexed and continuous multi-axis toolpaths, including better handling of tool orientation and collision avoidance.
8. **Save Post with File Option** — Users could now save the post-processor settings with the part file (this became even more refined in later releases).

Key Takeaways

The **2021 release** was a **usability and stability-focused update**. The biggest wins were:

- Dramatically improved user interface (re sizable dialogs)
- New dedicated **Saw Cutting** operation
- Better Knowledge Base functionality
- Faster performance across the board
- Continued improvements to multi-axis capabilities

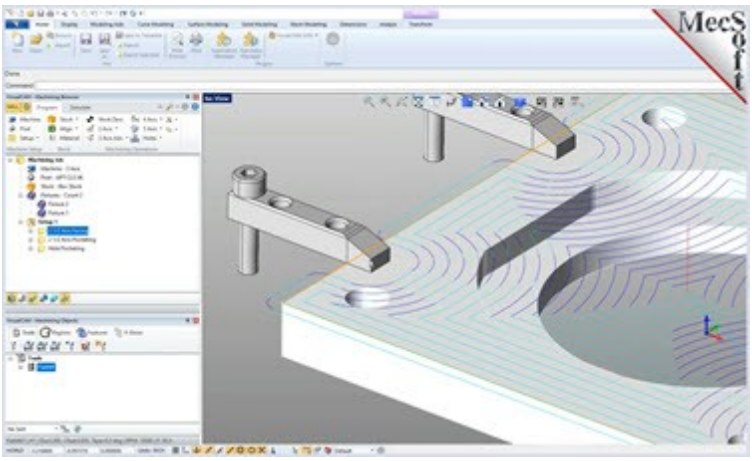
This release laid important groundwork for the more powerful automation features that came in 2022 and beyond.

7.7 What's New in 2020

Link: https://www.youtube.com/watch?v=fFTtyiQQGNA

This is the official “What’s New” webinar for the 2020 release of RhinoCAM and VisualCAD/CAM. It introduces several major new capabilities, including the G-Code Editor Module, Programmable Post Processors, significant Knowledge Base improvements, and various usability and performance enhancements.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2020 release
3:45	G-Code Editor Module (New – AMS Exclusive)	Full integration of back-plotting and simulation of actual posted G-code
12:30	Programmable Post Processors (Python-based)	New Python post processor framework for advanced customization
20:15	Knowledge Base Enhancements	Improved saving, loading, and new automation features
27:40	Usability Improvements	Re-sizable dialogs, new icons, better workflow
33:10	2.5-Axis & 3-Axis Enhancements	New operations and improvements in milling
39:50	4-Axis & 5-Axis Improvements	Enhanced multi-axis capabilities
45:20	Turning Module Updates	Improvements in the TURN module
50:10	Performance & Licensing Changes	Faster calculations and new licensing system
56:30	Live Demos & Q&A	Full walkthrough of major new features
1:04:00	Conclusion & Summary	Final recommendations

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the 2020 release:

1. **G-Code Editor Module is a Major Leap Forward** — For the first time, AMS subscribers could load the actual posted G-code and simulate it directly on the part geometry with full cut material simulation. This was a game-changer for catching post-processor errors before sending code to the machine.
2. **Programmable Post Processors (Python)** — 2020 introduced the Python-based programmable post processor framework. This gave advanced users and post developers unprecedented flexibility to create custom output for complex machines (drill banks, dual-head machines, special formats, etc.).
3. **Knowledge Base Improvements** — Significant enhancements were made to how Knowledge Bases are saved and loaded, including better support for complex multi-operation workflows and improved automation capabilities.
4. **Re-sizable Dialogs** — Almost every dialog in the software became re-sizable. This was one of the most requested usability improvements and made working on high-resolution monitors much more comfortable.
5. **Save Post with File Option** — Users could now save the post-processor settings with the part file, ensuring the correct post is automatically loaded when the file is opened on any computer.
6. **Performance Improvements** — Toolpath calculation and simulation were noticeably faster, especially on complex jobs with many operations.
7. **New Licensing System** — The 2020 release introduced improvements to the licensing infrastructure (Network Lock enhancements and better diagnostics).

Key Takeaways

The 2020 release was a foundational and power-user focused update. The two biggest highlights were:

1. Introduction of the G-Code Editor Module (AMS exclusive) — Allowing users to verify the *actual* posted G-code with full simulation.
2. Programmable Post Processors using Python — Opening the door for highly customized and advanced post-processing.

Other improvements (re-sizable dialogs, Knowledge Base enhancements, Save Post with File, and performance gains) made the software more modern and user-friendly.

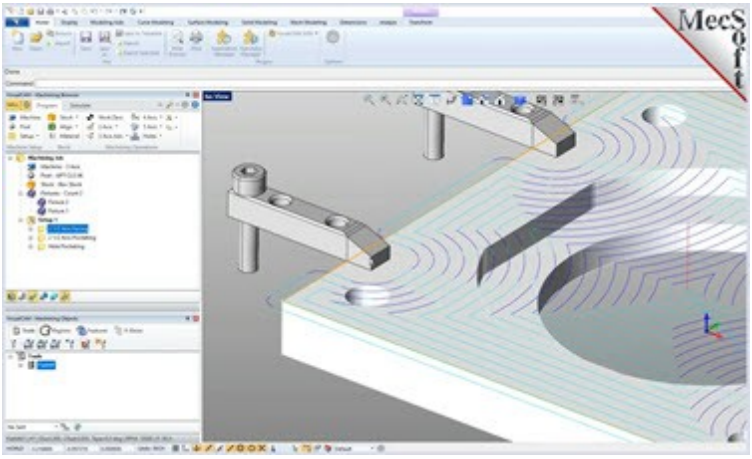
This release laid important groundwork for the even more powerful automation features that arrived in 2021 and 2022.

7.8 What's New in 2019

Link: https://www.youtube.com/watch?v=wpgUzTyS6MU

This is the official "What's New" webinar for the **2019 release** of VisualCAD/CAM and RhinoCAM. It covers major CAD improvements, new modules (Mill-Turn Beta), powerful new operations (Drag Knife, Cornering Actions, Clustering), Feature-Based Machining enhancements, simulation improvements, and significant Nesting Module upgrades.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction and Housekeeping	Overview of the 2019 release
4:00	VisualCAD 2019 Enhancements	Fit View to Selected, Undo/Redo View, Vertex Snap for meshes, High-Res Display support, improved translators
11:20	Mill-Turn Module (Beta)	New module for simultaneous milling + turning on 2-axis lathes
16:20	Turn Module Enhancements	Automatic exclusion of mill features using "sponge solid" technique
18:20	Mill Module Usability Enhancements	Cut depth per tool in library, tool number conflict check, title bar info display

Time	Topic / Section	Description
24:00	Feature-Based Machining Enhancements	Silhouette features + automatic feature detection and machining
28:50	Drag Knife Cutting Operation	New 2-axis operation for drag knives with automatic swivel & corner limit control
32:00	Cornering Actions	Loop, Sharp, Sharp Limited, Dog bone, and T-bone options for profiling
36:30	Clustering (Inside-Out Cutting)	New option to keep parts intact within the sheet during nesting/profiling
38:30	Facing Operation Enhancements	Automatic stock extent detection (no manual region selection needed)
39:50	Pocketing & High-Speed Cut Patterns	Trichoidal (high-speed) motion support
42:00	Simulation Enhancements	Shank/holder collision detection (highlighted in red)
43:50	3-Axis Toolpath Enhancements	Clear flats with fillets/chamfers, undercut connections in parallel finishing, faster pencil tracing
47:40	Skin Clearance Enhancements	3D geometry support for smarter retract heights
50:00	Post-Processor Integration	Auto-load max spindle/feed rates from post file
51:30	5-Axis Enhancements	New “Nutating” head support, world coordinate variables, user-defined drilling tools
54:30	Nesting Module Enhancements	Solid model nesting, Explode Cabinet Design utility, improved sheet definition
1:01:20	Q&A and Release Information	Final questions and December 2018 release details

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the **2019 release**:

1. **Feature-Based Machining + Silhouette Features** — One of the biggest productivity wins. The software can now automatically detect pockets, slots, holes, chamfers, **and silhouettes**, then generate complete toolpaths with a single command.
2. **Drag Knife Operation is Excellent for Sign Making** — New dedicated operation with automatic swivel control and corner limit angle settings — perfect for vinyl, fabric, and thin material cutting.
3. **Cornering Actions Give Precise Control** — New options (Dog bone, T-bone, Loop, Sharp Limited) allow you to control exactly how the tool behaves at internal and external corners during profiling.
4. **Clustering (Inside-Out Cutting)** — Prevents parts from separating too early during nested profiling by cutting from the inside out. Very useful for keeping small parts attached to the sheet until the very end.
5. **Automatic Stock Extent in Facing** — Facing operations now automatically use the stock silhouette, eliminating the need to manually select containment regions in most cases.
6. **Trichoidal (High-Speed) Pocketing** — New cut pattern option for smoother, faster, and more tool-friendly pocketing, especially in harder materials.
7. **Shank/Holder Collision Detection** — Simulation now clearly highlights collisions with the tool shank and holder in red and marks the operation as "dirty" so you know it needs review.
8. **Clear Flats with Fillets/Chamfers** — The "Clear Flats" option now properly handles parts with fillets and chamfers without gouging.
9. **Faster Pencil Tracing** — Up to 4–5× faster computation with a new "minimum pet length" parameter.
10. **Skim Clearance Now Supports 3D Geometry** — Retract heights are calculated more intelligently when 3D features are present, reducing unnecessary Z moves.
11. **Nesting Solid Models + Explode Cabinet Utility** — You can now nest actual 3D solids, surfaces, and meshes. The new "Explode Cabinet Design" utility automatically flattens 3D cabinet assemblies into 2D panels ready for nesting — a huge time saver for cabinet shops.
12. **Save Post with File** — A major usability improvement that lets you lock the correct post-processor to each job file.

Key Takeaways

The **2019 release** was a **very strong, feature-rich update** that focused on **automation, usability, and real-world productivity**.

Biggest highlights:

- Feature-Based Machining + Silhouette detection
- New Drag Knife and Cornering Actions operations
- Major Nesting Module upgrades (solid model nesting + cabinet explosion utility)
- G-Code Editor (introduced in 2020, but the foundation was laid here)
- Much better simulation and collision detection

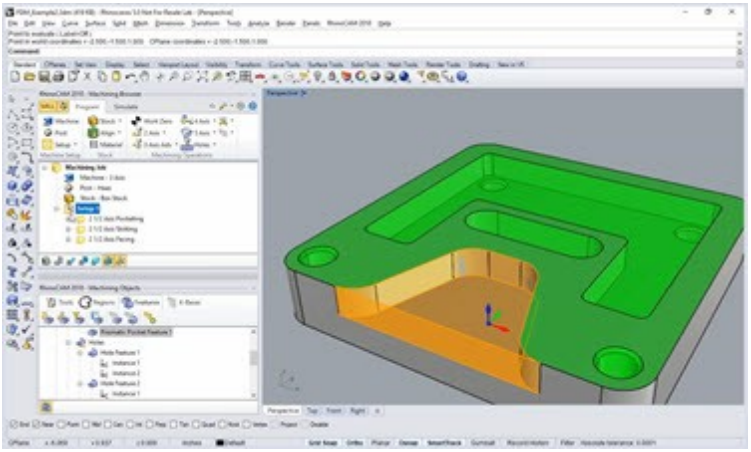
This was one of the most impactful releases in the MecSoft lineup at the time.

7.9 What's New in 2018

Link: https://www.youtube.com/watch?v=-rNGFqlJ04E

This is the official “What’s New” webinar for the 2018 release of MecSoft CAM (VisualCAD/CAM and RhinoCAM). It introduces several groundbreaking features, most notably Automatic Feature Detection & Machining (AFD/AFM), the new Mesh Module, significant usability improvements, and a major change making the ART and NEST modules free with all Mill configurations.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2018 release
4:00	Automatic Feature Detection (AFD)	Automatically identifies pockets, holes, slots, chamfers, and planar faces on solid models

Time	Topic / Section	Description
11:00	Automatic Feature Machining (AFM)	One-click toolpath generation using Knowledge Bases
18:00	Interactive Feature Machining	Right-click detected features to choose operations
24:00	VisualCAD/CAM 2018 Usability Improvements	Chain selection, pre-selection highlighting, ghosted display, surface selection without exploding solids
32:00	2.5-Axis Enhancements	Chamfer height/split options, engraving order sorting, bridge/tab feed rate reduction, new Filleting operation
39:00	3-Axis Enhancements	"Follow Containment", SteepZ top/bottom containment, region sorting
45:00	4-Axis & 5-Axis Enhancements	"Create Round" operation, helical cut pattern, locking one rotary axis in 5-axis
51:00	New Mesh Module	Point cloud to mesh, Boolean operations, decimation, smoothing, auto-repair
57:00	Simulation & Collision Enhancements	Shank/holder collision detection with red highlighting
1:02:00	Peck Tapping & Feeds/Speeds	New Peck Tapping operation + max spindle/feed limits
1:08:00	Licensing & Module Changes	ART and NEST modules now free with all Mill configurations
1:15:00	RhinoCAM API Introduction	New Python/.NET API for automation
1:20:00	Q&A and Year-End Sale Announcement	Final questions and promotional details

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the 2018 release:

1. **Automatic Feature Detection & Machining (AFD/AFM) is the Star Feature** — This was the biggest addition in 2018. After creating a Knowledge Base, you can detect all machinable

features on a solid model (in all orientations) and generate complete toolpaths with one click. This dramatically reduces programming time on prismatic parts.

2. **Interactive Feature Machining** — Even if you don't use full automation, you can right-click any detected feature and instantly choose the operation type (pocketing, profiling, drilling, etc.) with parameters already populated.
3. **ART and NEST Modules Are Now Free** — One of the most impactful changes: the ART (artistic relief) and NEST (nesting) modules became included at no extra cost with all Mill configurations starting in 2018.
4. **Ghosted Display Mode** — A very useful new display option that makes hidden features (like internal cavities) visible without having to hide the outer geometry.
5. **Chain Selection (Shift + Click)** — You can now quickly select chains of curves or surface edges by holding Shift — a major time saver when selecting containment or profiling geometry.
6. **Holder Collision Detection** — Simulation now detects and highlights collisions with the tool shank and holder (shown in red) and marks the operation as "dirty" so you know it needs attention.
7. **"Create Round" 4-Axis Operation** — A simple but powerful new operation that automatically turns rectangular stock into a cylinder using the 4th axis — very useful for preparing stock for round parts.
8. **Follow Containment in Horizontal Finishing** — Prevents unnecessary retracts by making the tool follow the containment boundary instead of retracting and re-engaging.
9. **Peck Tapping Operation** — New dedicated operation that allows you to specify peck depth for tapping deep holes.
10. **RhinoCAM API** — The introduction of a Python/.NET API opened the door for custom automation and integration with other systems.

Key Takeaways

The 2018 release was one of the most significant in MecSoft's history because it introduced Automatic Feature Detection & Machining (AFM) — a feature that fundamentally changed how users program prismatic parts.

Biggest highlights of 2018:

- Automatic Feature Detection + One-Click Machining (AFM)
- ART and NEST modules made free
- New Mesh Module
- Major usability improvements (chain selection, ghosted display, re sizable dialogs)
- Holder collision detection in simulation

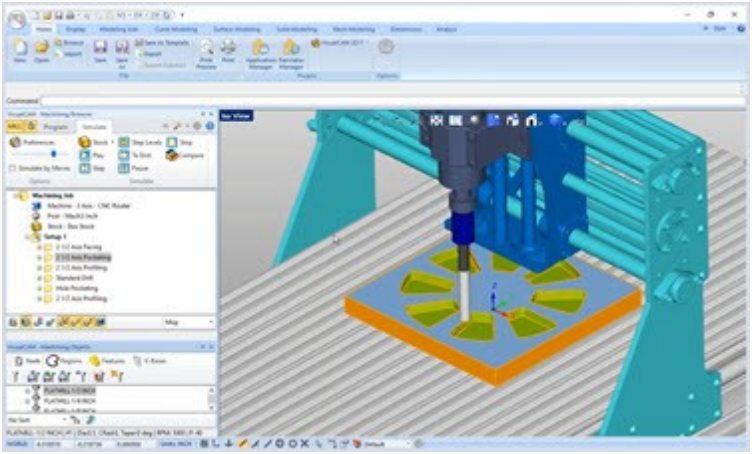
This release laid the foundation for the even more advanced automation features that came in later years (especially 2019 and beyond).

7.10 What's New in 2017

Link: https://www.youtube.com/watch?v=J-sjMZ1pcqg

This is the official “What’s New” webinar for the 2017 release of RhinoCAM and VisualCAD/CAM. It introduces several important new capabilities, most notably the first version of Automatic Feature Detection, major 5-axis improvements, Knowledge Base enhancements, and significant performance and usability upgrades.

Source Files: Refer to the CAMJam 2025 Files Archive for Matching Source Files



Topic Index with Timestamp Links

Time	Topic / Section	Description
0:00	Introduction & Agenda	Overview of the 2017 release goals
3:30	Automatic Feature Detection (AFD)	First version of automatic feature recognition on solid models
10:00	Automatic Feature Machining (AFM)	Using Knowledge Bases to automatically machine detected features
17:00	5-Axis Enhancements	New Swarf Machining, Flow Curve, and improved tool orientation
24:00	Knowledge Base Improvements	Better saving, loading, and management of Knowledge Bases
30:00	2.5-Axis & 3-Axis Enhancements	New operations and improved toolpath strategies
37:00	Usability & Performance Improvements	Faster calculations, better display options, and workflow enhancements

Time	Topic / Section	Description
44:00	Turning Module Updates	Improvements to the TURN module
50:00	Post-Processor & Licensing Changes	Updates to post processing and licensing system
55:00	Live Demonstrations	Full walkthrough of major new features
1:05:00	Q&A and Conclusion	Final questions and summary

Key Tips & Best Practices Extracted

Here are the most valuable new features and practical takeaways from the 2017 release:

1. **Automatic Feature Detection (AFD) was the biggest new feature** — This was the very first version of what later became Automatic Feature Machining (AFM). It could automatically detect pockets, holes, slots, chamfers, and planar faces on solid models — a major step toward reducing manual geometry selection.
2. **Knowledge Bases + Feature Detection = Powerful Automation** — Even in its first version, combining Knowledge Bases with Automatic Feature Detection allowed users to program many prismatic parts with significantly less manual work.
3. **Swarf Machining Introduced** — The new Swarf operation (keeping the side of the tool in contact with a ruled surface) was a major addition for 5-axis users, especially in mold and aerospace work.
4. **Flow Curve Operation** — A new 5-axis strategy that allows the tool to follow a user-defined curve while staying normal to the surface — very useful for complex organic shapes.
5. **Improved Knowledge Base Management** — Better tools for saving, organizing, and loading Knowledge Bases made automation more practical for daily use.
6. **Performance Improvements** — Toolpath calculation and simulation were noticeably faster, especially on complex 3-axis and 5-axis jobs.
7. **Usability Enhancements** — Several small but meaningful improvements were made to the interface and workflow (better icons, display options, and dialog behavior).
8. **Turning Module Refinements** — Several behind-the-scenes improvements were made to the TURN module for better reliability and output quality.

Key Takeaways

The 2017 release marked the beginning of MecSoft's push toward intelligent automation. The introduction of Automatic Feature Detection was a foundational change that set the stage for the much more advanced Feature-Based Machining capabilities that came in 2018 and beyond.

Biggest highlights of 2017:

- First version of Automatic Feature Detection (AFD)
- New 5-axis operations: Swarf and Flow Curve
- Improved Knowledge Base functionality
- Better performance and usability across the board

This release was an important stepping stone that made MecSoft CAM significantly more powerful and user-friendly.

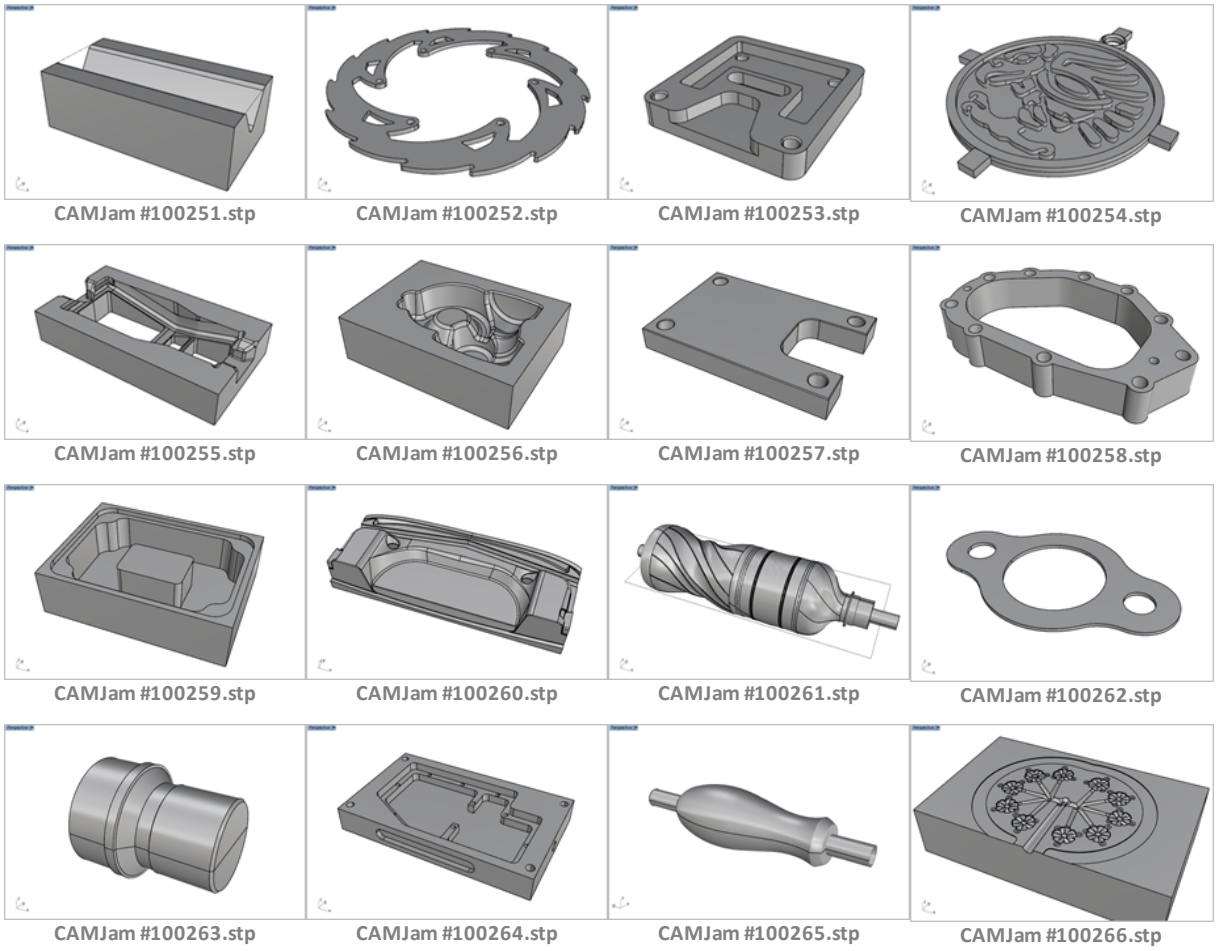
Practice File Archive

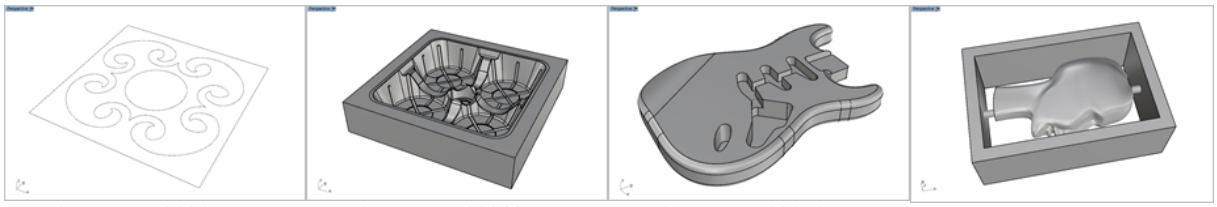
The following files are used as examples in the CAMJam video archive. The files are available in the following CAD neutral formats. They can be opened in any version of each of the CAD platforms supported by the MecSoft CAM plug-ins (VisualCAD/CAM, Rhino3D). Additional practice files are also included in the archive.

File Formats

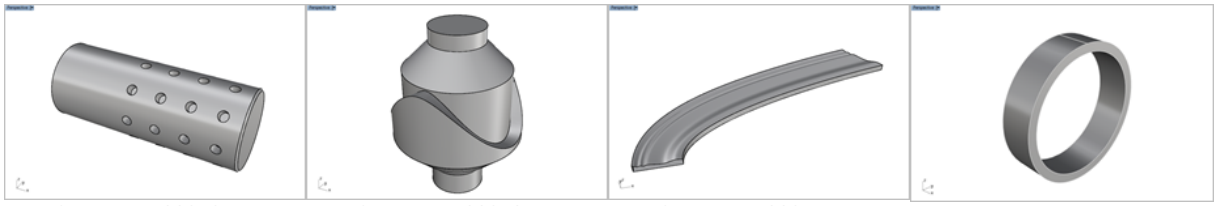
- *.STP (3D Solid Models)
- *.IGS (3D Surface Models)
- *.STL (3D Mesh Models)
- *.DWG (2D Drawings)

File Archive Contents



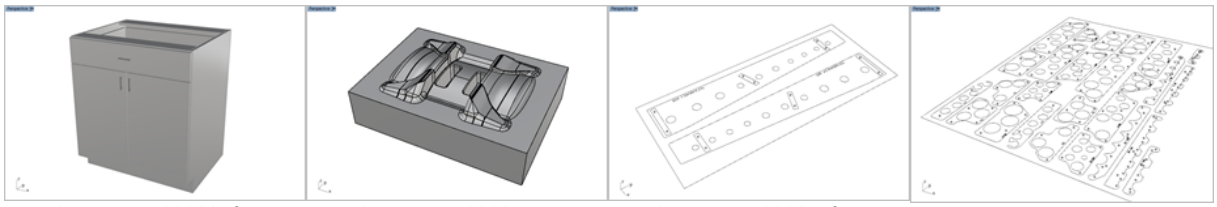


CAMJam #100267.stp CAMJam #100268.stp CAMJam #100270.stp CAMJam #100271.stp

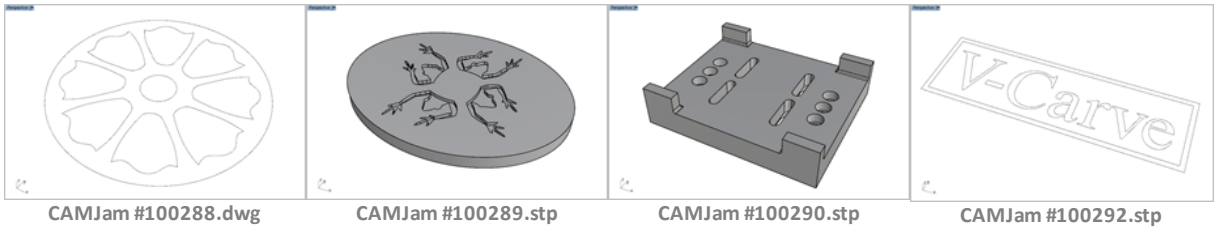


CAMJam #100272.stp CAMJam #100273.stp CAMJam #100274.stp CAMJam #100276.stp

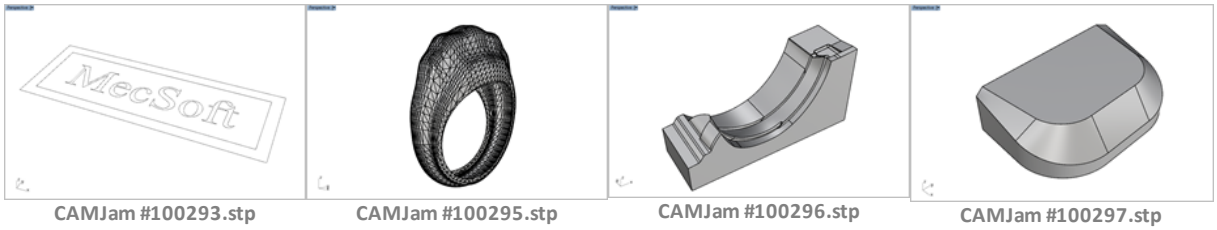
CAMJam #100278.stp CAMJam #100279.stp CAMJam #100280.stp CAMJam #100282.stp



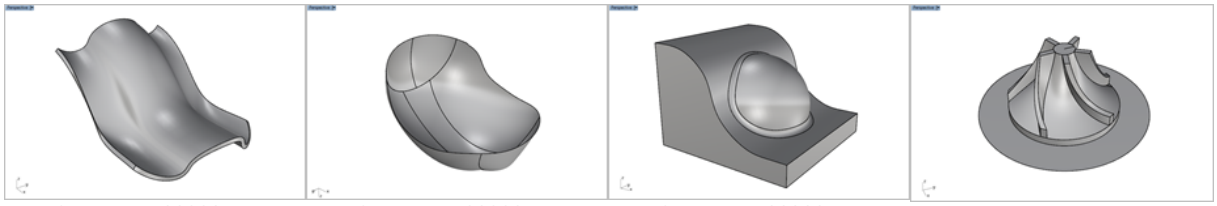
CAMJam #100283.skp CAMJam #100284.stp CAMJam #100285.dwg CAMJam #100286.dwg



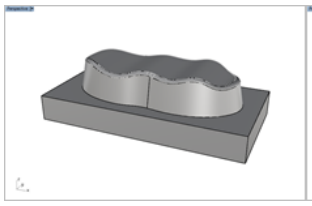
CAMJam #100288.dwg CAMJam #100289.stp CAMJam #100290.stp CAMJam #100292.stp

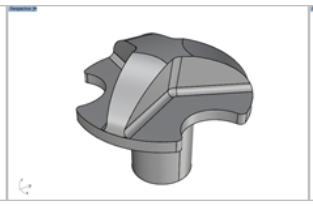


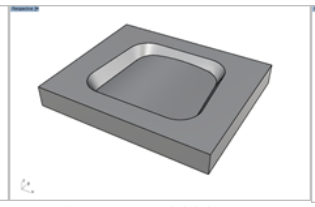
CAMJam #100293.stp CAMJam #100295.stp CAMJam #100296.stp CAMJam #100297.stp

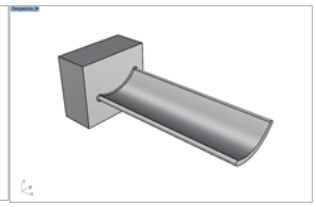


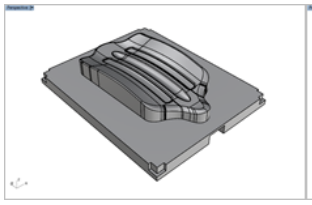
CAMJam #100298.stp CAMJam #100299.stp CAMJam #100300.stp CAMJam #100302.stp

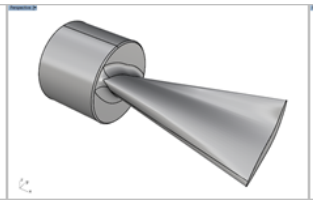

CAMJam #100303.stp

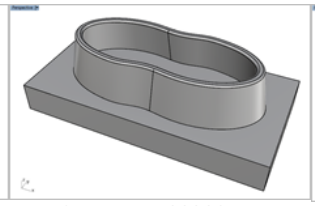

CAMJam #100304.stp

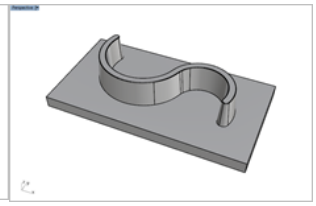

CAMJam #100305.stp

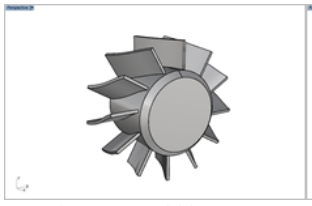

CAMJam #100306.stp

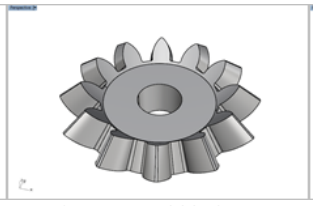

CAMJam #100307.stp

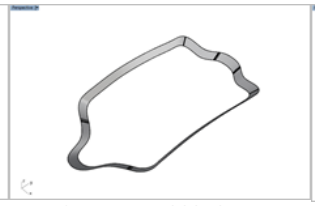

CAMJam #100308.stp

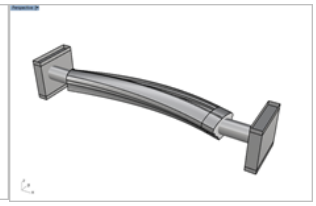

CAMJam #100309.stp

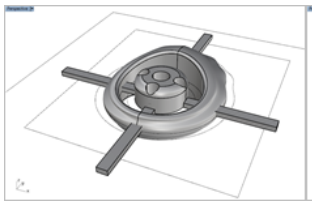

CAMJam #100310.stp

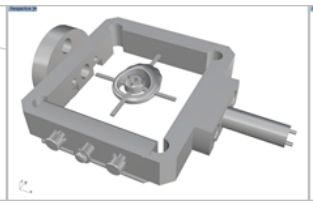

CAMJam #100311.stp

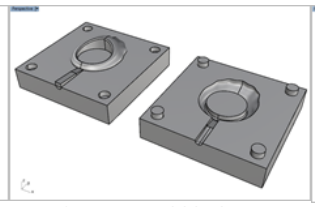

CAMJam #100312.stp

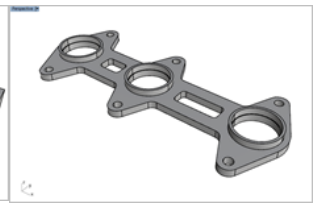

CAMJam #100313.stp

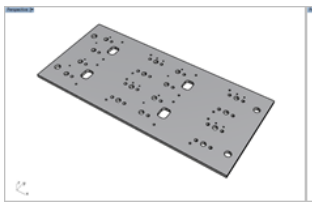

CAMJam #100314.stp

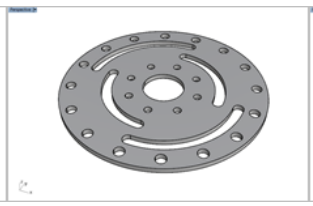

CAMJam #100315.stp

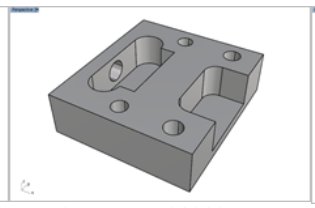

CAMJam #100315-stl


CAMJam #100316.stp

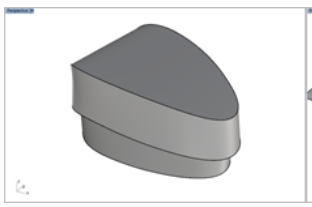

CAMJam #100319.stp

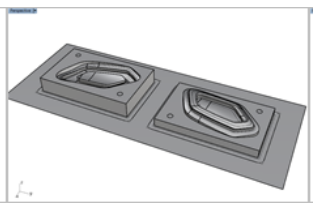

CAMJam #100320.stp


CAMJam #100321.stp

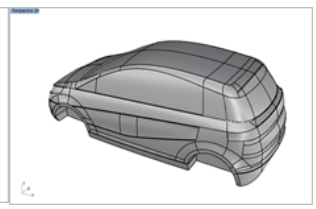

CAMJam #100322.stp


CAMJam #100323.stp

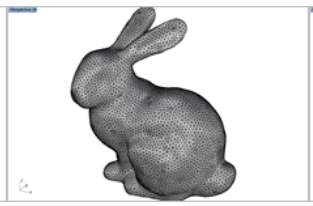

CAMJam #100324.stp

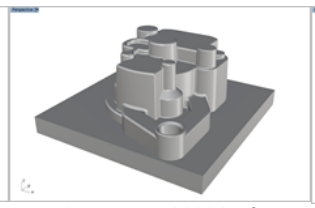

CAMJam #100324.stp

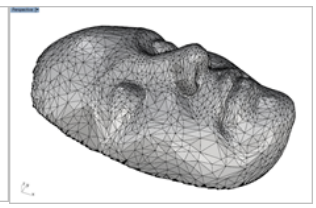

CAMJam #100326.stp


CAMJam #100327.stp

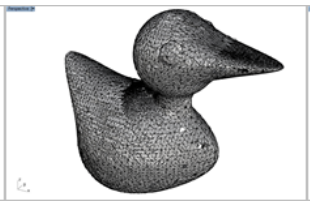

CAMJam #100329.stp


CAMJam #100331.stl

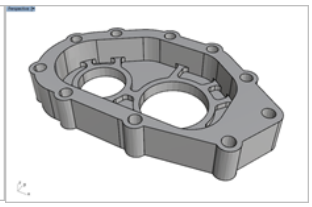

CAMJam #100330.stl


CAMJam #100332.stl

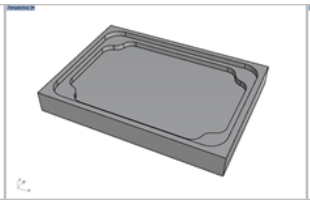

CAMJam #100333.stl

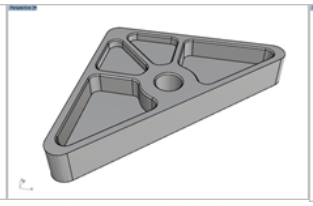

CAMJam #100334.stl

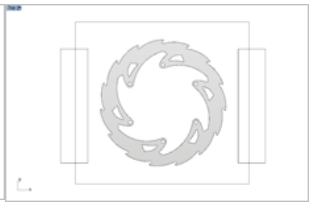
CAMJam #100334.dwg

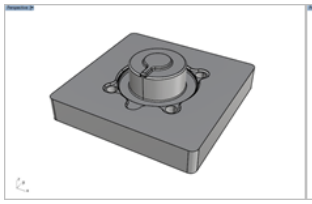

CAMJam #100338.stp

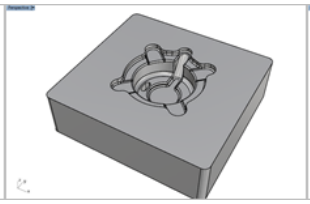

CAMJam #100340.stp

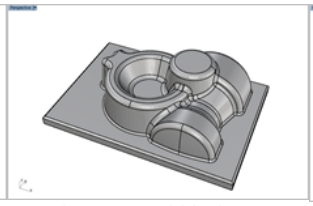

CAMJam #100341.stp

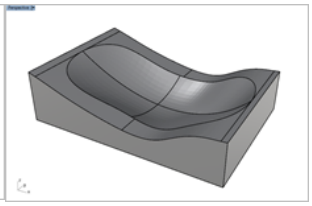

CAMJam #100342.stp

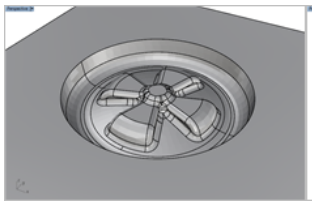

CAMJam #100343.dwg

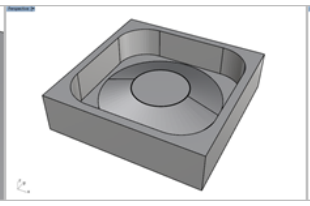

CAMJam #100344.stp


CAMJam #100345.stp

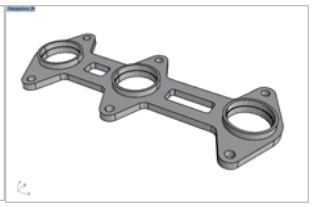

CAMJam #100346.stp


CAMJam #100347.stp

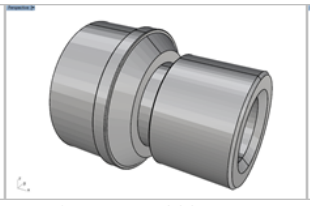

CAMJam #100348.stp

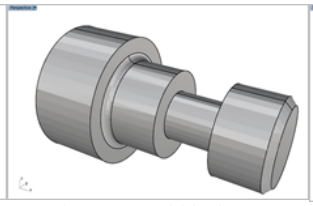

CAMJam #100349.stp

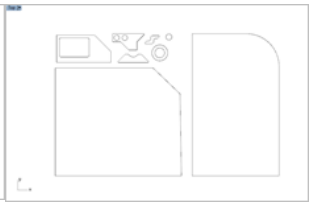

CAMJam #100350.stl


CAMJam #100351.stp


CAMJam #100354.stp


CAMJam #100355.stp


CAMJam #100356.stp


CAMJam #100357.dwg

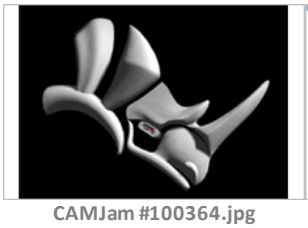

CAMJam #100359.jpg

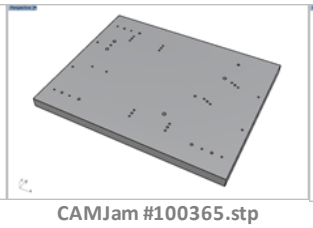

CAMJam #100360.stl


CAMJam #100361.stl

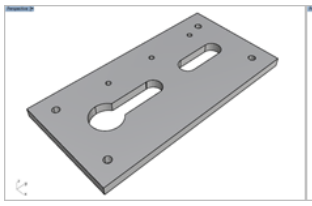

CAMJam #100362.bmp

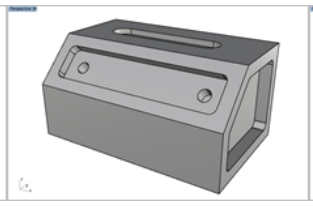

CAMJam #100363.jpg

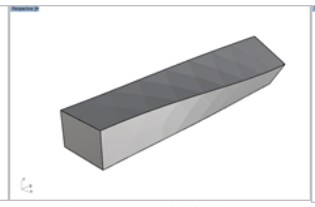

CAMJam #100364.jpg


CAMJam #100365.stp

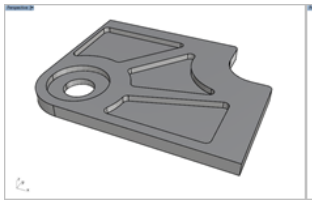

CAMJam #100366.stl

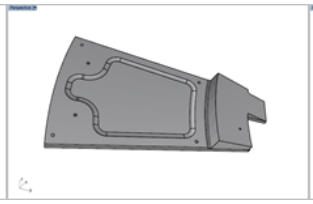

CAMJam #100367.stp

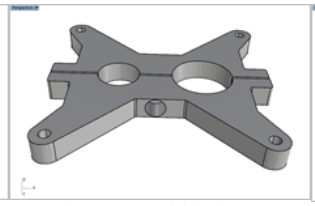

CAMJam #100368.stp


CAMJam #100369.stp


CAMJam #100336.dwg

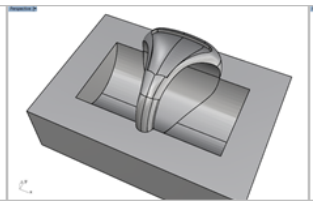

CAMJam #100370.stp


CAMJam #100371.stp

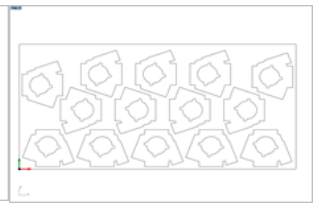

CAMJam #100373.stp

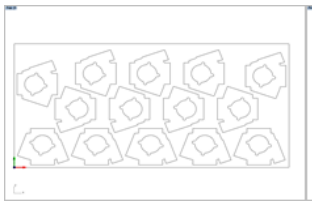

CAMJam #100375.stp

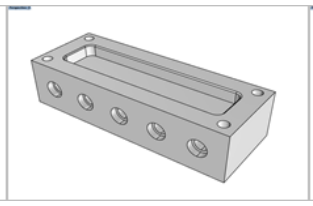

CAMJam #100376.stp

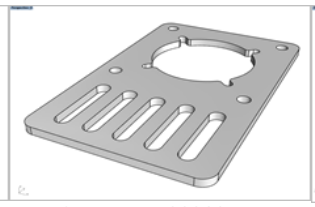

CAMJam #100377.stp

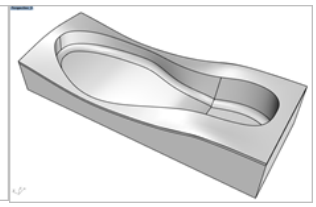

CAMJam #100378.stp

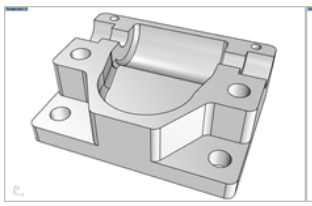

CAMJam #100381.dwg

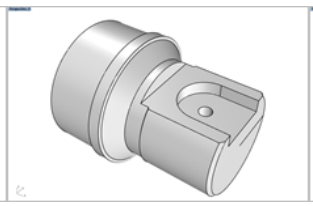

CAMJam #100381.dwg

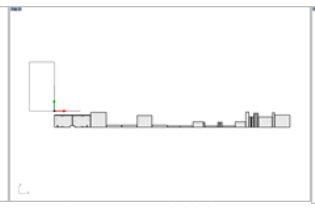

CAMJam #100382.stp

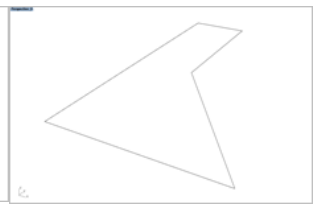

CAMJam #100383.stp


CAMJam #100384.stp

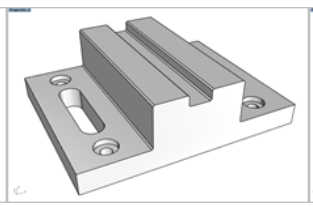

CAMJam #100385.stp

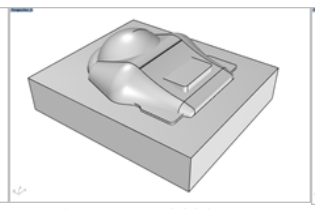

CAMJam #100386.stp

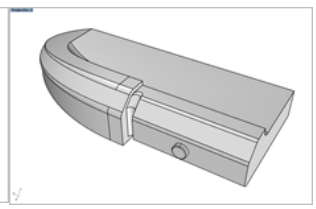

CAMJam #100387.stp

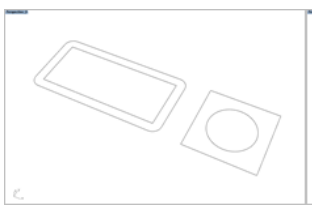

CAMJam #100388.stp

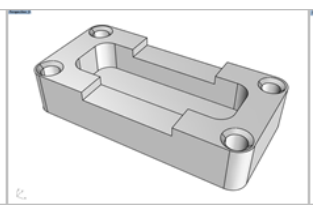

CAMJam #100389.dwg

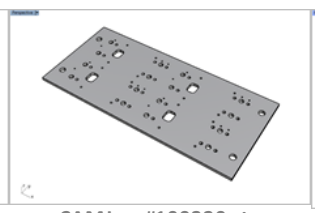

CAMJam #100390.stp

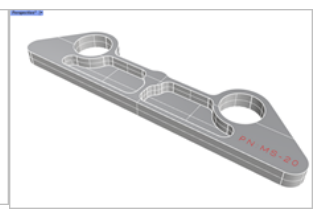

CAMJam #100391.stp

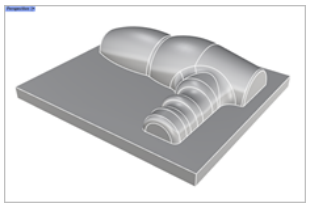

CAMJam #100392.stp


CAMJam #100393.dwg


CAMJam #100394.stp


CAMJam #100320.stp


CAMJam #100443.stp



CAMJam #100449.stp

Case Studies

The table below lists many case studies by users who have demonstrated their success with MecSoft CAM products. The icons and columns indicate which configuration was used and a link is provided to the case study or tech blog. These studies will provide you with inspiration and guidance as you implement your MecSoft product.

User Case Studies and Tech Blogs									
Case Study/Blog	STD	XPERT	PRO	PREM	Rhino	VCC	CAD	ART	NEST
Doran Oster									
SITU									
Piedmont Composites									
RustFab									
Composite Solutions									
Rhodes Yacht									
Chase Boats									
Dovetailers									
Curtis Erpelding Furniture									
Power by Design									
CNC Masters									
General Fence									
Conley Manufacturing									
Lane Manufacturing									

User Case Studies and Tech Blogs									
Green Fuel	✓					✓	✓		
KOZM Guitars	✓					✓	✓		
Insight Exhibits	✓				✓				
KITEX SIA	✓					✓	✓		
LMN Architects	✓				✓				
OESH Shoes	✓					✓	✓		
Dubes Customs	✓					✓	✓		
Danish Art Workshop	✓				✓				
Quail Pens		✓			✓				
Blase Photography			✓		✓				
Bill Bancroft Furniture			✓		✓				
AirMotive Specialties			✓		✓				
Pete Sorenson	✓				✓				
Mike Wasinski			✓		✓				
Institute for Biodigital Architecture & Genetics	✓				✓				
Louisiana Tech	✓				✓				
McCafferty Dulcimers	✓				✓				

User Case Studies and Tech Blogs									
Hancock High School									
Designer Grains									
RC B1 Lancer Bomber Project									
PolyFab									
Oskayak High School									
IYRS									
ENVIBOATS									
Duna									
Pedalino Bicycles									
Chris Dungy Cello Makers									

Index

- 2 -

- 3 -

- 4 -

- 5 -

- A -

- C -

- F -

- G -

- H -

- I -

- J -

- K -

- L -

- M -

- N -

- O -

- P -

- S -

- T -

- V -

- W -