MecSoft Corporation
Your CAM Partner

What's New in VisualCAM 2020 for SOLIDWORKS

Jan 27, 2020

This document describes new features and enhancements introduced in MecSoft's VisualCAM for SOLIDWORKS product.
NOTE: This information is provided in good faith but MecSoft reserves the right to modify or omit any of the items described in this document when the product is released.

© 2020, MecSoft Corporation

CONTENTS

VisualCAM 2020 4
G-Code editor Module 5
Modes of Operation 6
Profile-Nest Module 9
Mill-Turn Module 12
Common CAM Enhancements 15
Licensing Enhancements 15
What's new in VisualMILL 2020 15
Usability Enhancements 15
2-Axis Enhancements 17
3-Axis Enhancements 20
4 Axis Toolpath Enhancements 21
5 Axis Toolpath Enhancements 21
Drilling Enhancements 21
Tooling Enhancements 22
Feeds/Speeds Enhancements 23
Simulation Enhancements 24

© 2020, MecSoft Corporation

Machine Tool Simulation Enhancements 25
Knowledge Base Enhancements 25
Post Processor Enhancements 25
Bugs Fixed 26
What's new in VisualTURN 2020 26

© 2020, MecSoft Corporation

This document describes the new functionality that is being introduced with the release of the VisualCAM 2020 for SOLIDWORKS product. This document is organized by listing and describing each of the enhancements incorporated into each of the constituent modules of VisualCAM.

VISUALCAM 2020

VisualCAM 2020 is a plug-in that runs inside the SOLIDWORKS Design product and hosts the following modules:

1. MILL
2. TURN
3. GCODE EDITOR (new in 2020)
4. PROFILE-NEST (new in 2020)
5. MILL-TURN (new in 2020)

Each of these modules can be licensed and invoked separately of the other modules. This section describes the various enhancements and improvement to each of the modules.

Licensing Information for the new modules

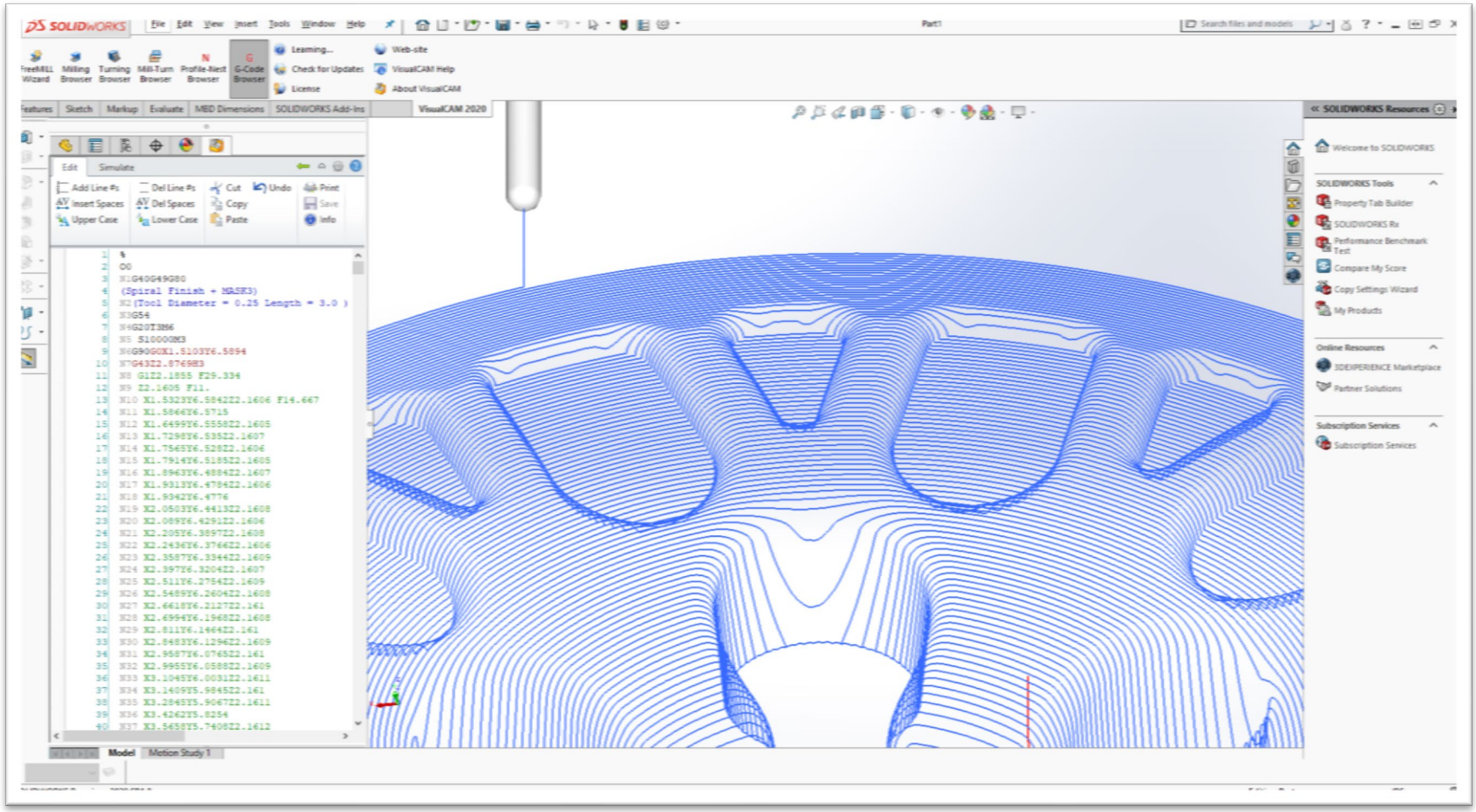
1. **Annual Maintenance SUBSCRIPTION (AMS) Customers of VisualCAM PRO & PRE Configurations will get a license of all three new (MILL-TURN, GCODE-EDITOR and PROFILE-NEST) modules free of cost**

2. **Annual Maintenance SUBSCRIPTION (AMS) Customers of VisualCAM STD & EXP Configurations will get a license of both the GCODE-EDITOR and PROFILE-NEST modules free of cost**

© 2020, MecSoft Corporation

G-CODE EDITOR MODULE

A new G-code editor module has been introduced in the 2020 product. A screen capture of this module with G-code being displayed is shown below.



This module allows various functions to be performed on the G-code files. These are described below.

© 2020, MecSoft Corporation

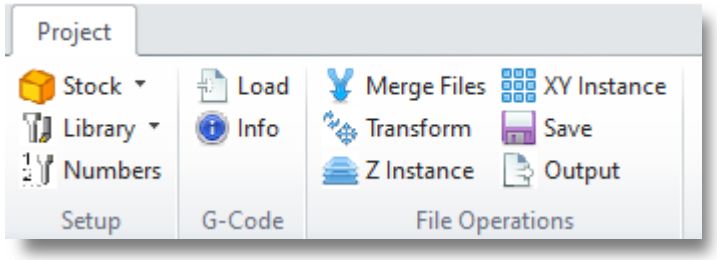
.

MODES OF OPERATION

There are two modes of operation of the G-Code editor module. These are as follows:

- Project Mode – This mode is active when commands to work with the various objects such as stock, tools as well as the loaded G-code files are needed. In this mode the main tree control will be displayed in the browser window.
- Editor Mode – This mode is active when commands to work with a single G-code file are necessary. This mode is launched by selecting a single G-code file in the main tree-control and then editing it by either double clicking this icon or by clicking the right mouse button and selecting the edit command. In this mode the actual G-code lines will be displayed in the browser window.

SETUP MODE



PROJECT TAB FUNCTIONS

1. Stock – allows the definition of the stock model for simulation:
   - Box stock, Part Box Stock, Cylinder Stock, Part Cylinder Stock
2. Tool Library– allows the loading of a tool library created in the CAM system
3. Tool Numbers – allows the definition of the tool crib to be used in simulation
4. Load – Allows loading of G-code file
5. Info – Overall information of selected or all loaded G-code files
6. Merge – Allows merging of selected G-code files into one
7. Transform – allows move, translate, rotate and mirror of G-code files
8. XY Instance – allows the 2 D instancing of selected G-code files
9. Z Instance – allows the Z instancing of selected G-code files
10. Save – Save all modified G-code files
11. Output – Output G-code files to an external program

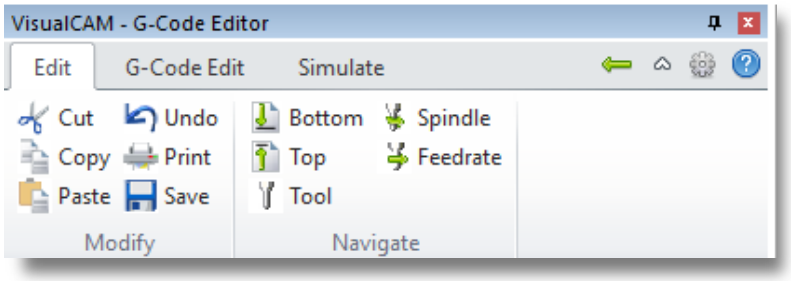
© 2020, MecSoft Corporation

EDIT MODE

In the edit mode, 3 tabs are available to the user. These are

- Edit Tab – Functions for text navigation and simple text editing
- G-code Edit Tab – Functions for editing the actual code
- Simulate Tab – Functions for simulation/back-plotting

EDIT TAB

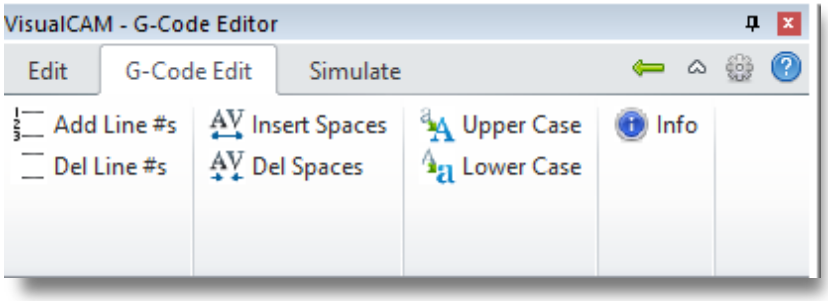


EDIT TAB FUNCTIONS

1. Cut – Cut the text selection
2. Copy – Copy the selected text to the clipboard
3. Paste – Paste the clipboard text back to the editor
4. Undo – Undo the previous edit operation
5. Print – Print the edited G-code file
6. Save – Save the edited G-code data to a file
7. To Bottom – Navigate to the bottom of the edit control
8. To Top – Navigate to the top of the edit control
9. To next tool change – Navigate to the next tool change statement
10. To next spindle– Navigate to the next spindle statement

© 2020, MecSoft Corporation

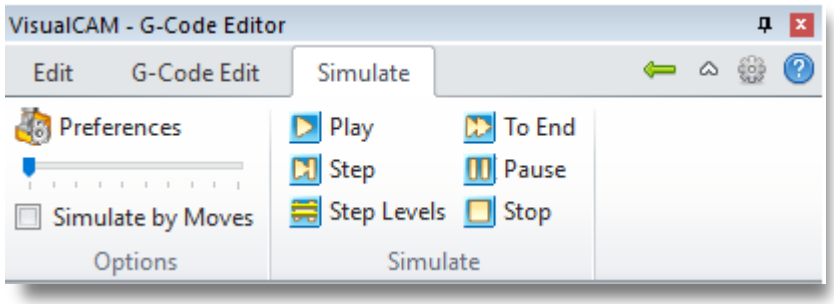
G-CODE EDIT TAB



G-CODE EDIT TAB FUNCTIONS

1. Add Line #s –Inserts line numbers to the G-code lines
2. Delete Line #s – Removes all line numbers from the G-code lines
3. Insert Spaces – This inserts spaces between G-code blocks in the file
4. Delete Spaces – This deletes any spaces between G-code blocks
5. Upper Case – This makes all text upper case in the file
6. Lower Case – This makes all text lower case
7. Info – This provides the statistics on the G-code file

SIMULATE TAB



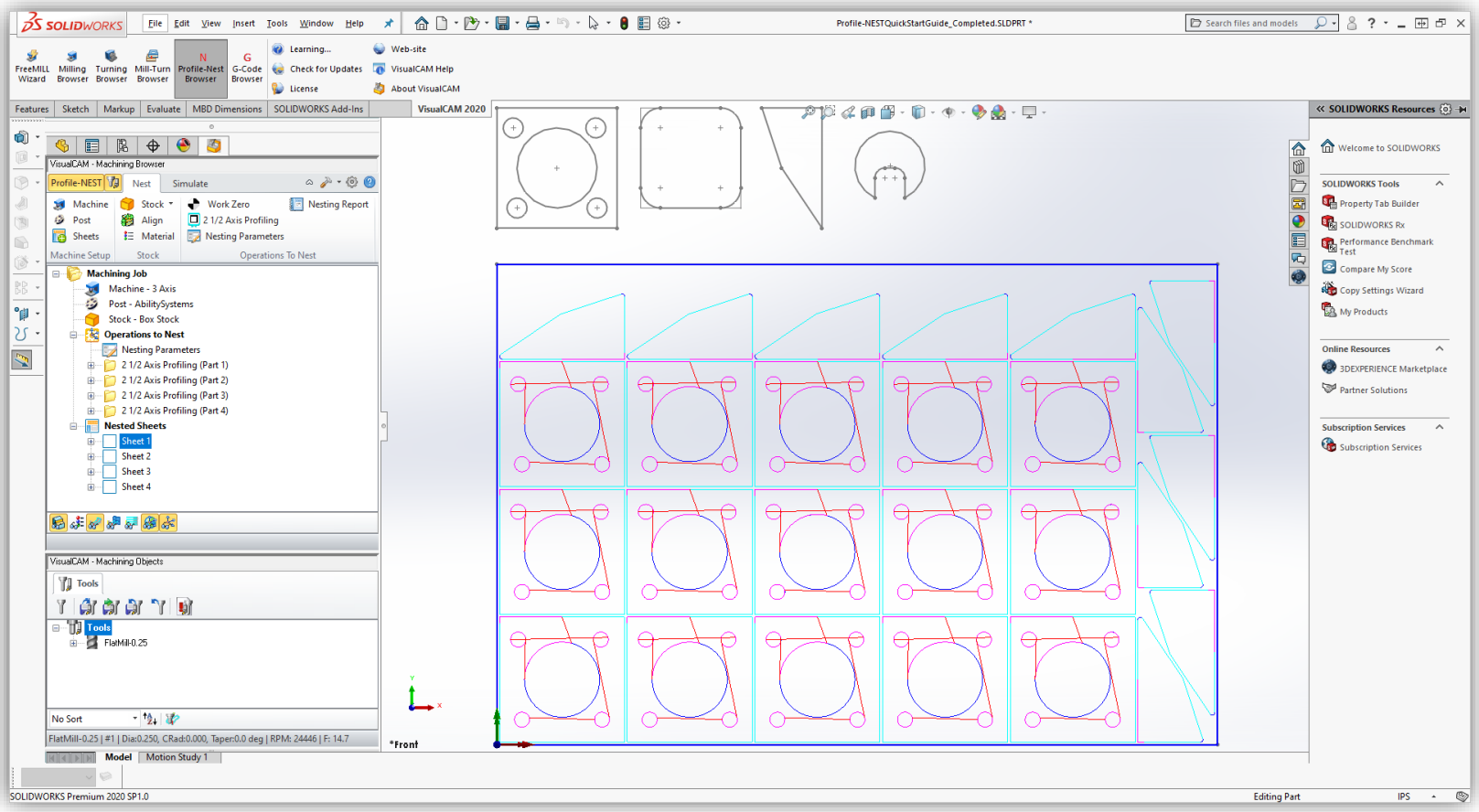
SIMULATE TAB FUNCTIONS

1. Play – Continuously loop through the simulation
2. Step by count – The count can be changed by using the slider bar or by setting the count in the options dialog
3. Step by levels – Step the simulation in levels if levels are detected in the toolpath
4. Simulate to End – Simulate the toolpath to the end
5. Pause the simulation
6. Stop – Stop the simulation

© 2020, MecSoft Corporation

PROFILE-NEST MODULE

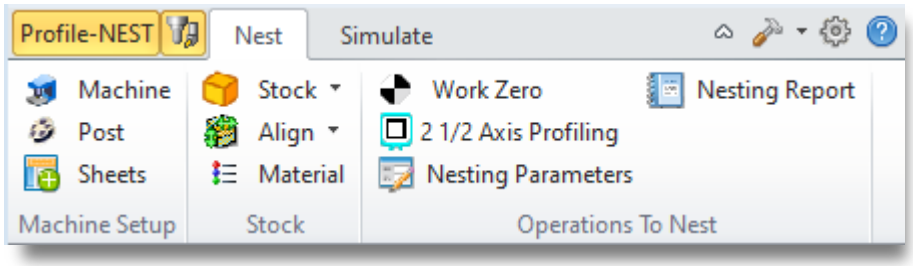
A new Profile-NEST module has been introduced in the 2020 product. This module allows nesting of 2 ½ Axis Profile toolpaths instead of just curves as the NEST module currently does. This module is tailored for the woodworking and flat panel machining industries. A screen capture of the UI is shown below.



The Profile-NEST module launches a new browser window which has the following two tabs:

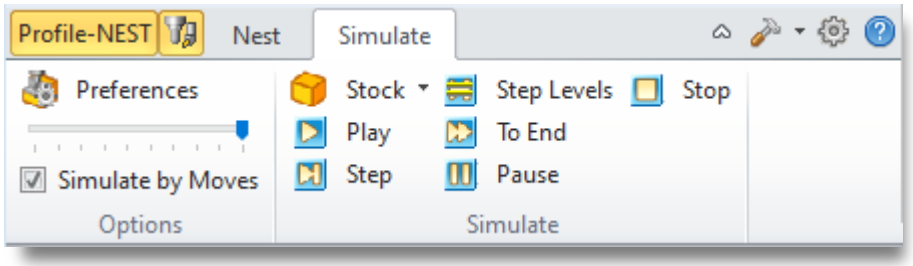
© 2020, MecSoft Corporation

- Nest – This tab is used to setup the sheets, the 2 ½ Axis Profiling operation to be nested, along with the various with other objects used in this module
- Simulate – This tab is used to run the toolpath simulation and verify the nesting results



NEST TAB FUNCTIONS

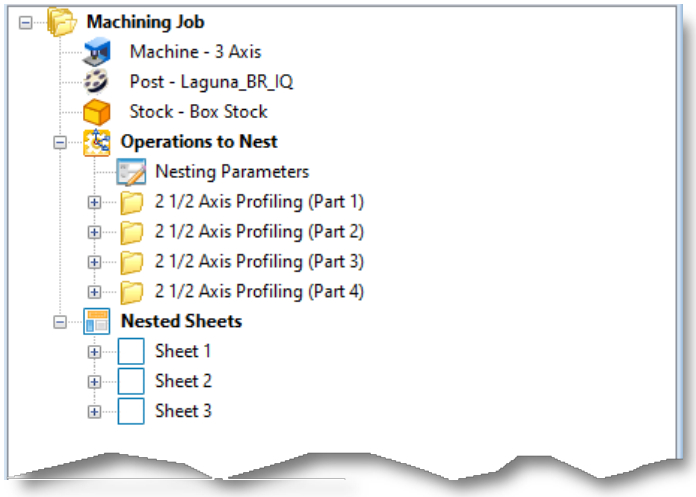
1. Machine – Defines machine tool (Same as MILL)
2. Post – Defines Post (Same as MILL)
3. Sheets – Define sheets to nest toolpaths in
4. Stock – Create stock model (Same as MILL)
5. Align – Align stock to toolpaths
6. Material – Select material for stock (Same as MILL)
7. Work Zero – Set work zero for operations (Same as MILL)
8. 2 ½ Axis Profiling – Create operations with count information
9. Nesting Parameters – Define parameters to nest
10. Nesting Report– Create nesting report



SIMULATE TAB FUNCTIONS

The simulation tab functions are the same as those found in the MILL module.

© 2020, MecSoft Corporation



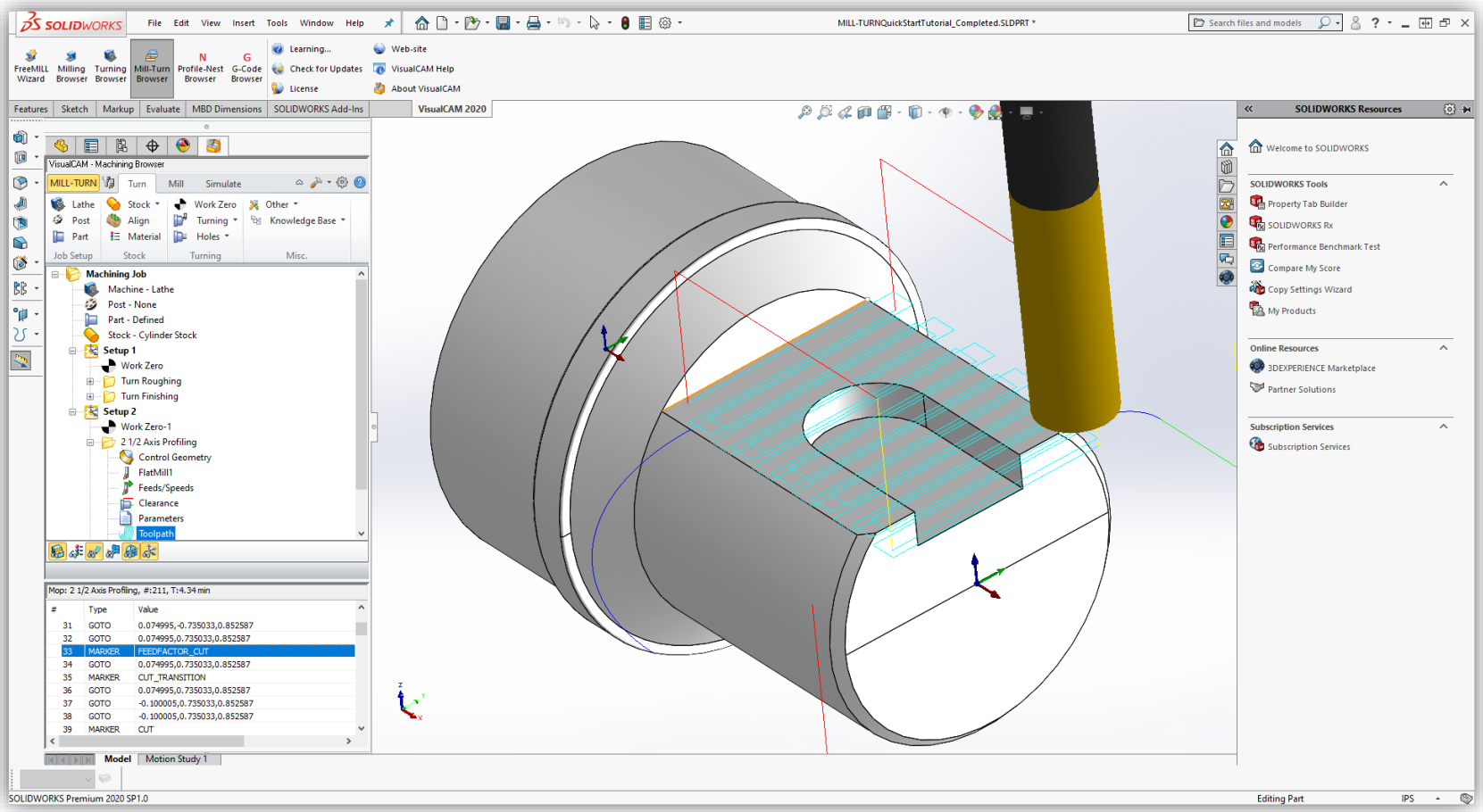
PROFILE NESTING WORKFLOW

1. User creates one or more 2 ½ Axis Profiling operations. These can be considered as parent operations. User specifies the count of each of these operations to be nested.
2. The user defines additional nesting parameters including sheets to use for nesting.
3. The system will compute the nested operations and display them in the browser window under the Nested Sheets icon as shown here.
4. Selecting and expanding each of the Sheet icons under the Nested Sheets icon will display all of the nested operations under this icon, as well as display them in the graphics window
5. Once the nested operations are created, they can be post-processed by selecting the Nested Sheets icon or each of the Sheet icons individually.

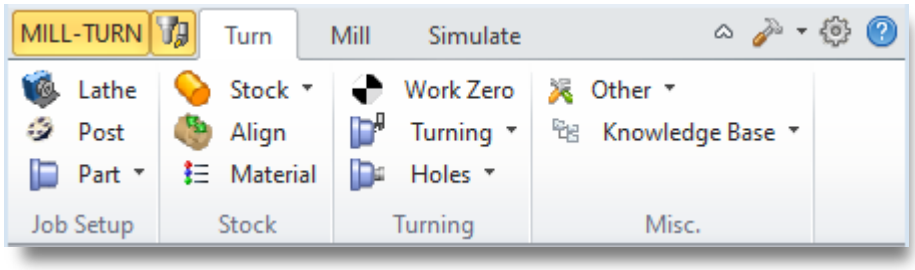
© 2020, MecSoft Corporation

MILL-TURN MODULE

A new Mill-TURN module has been introduced in the 2020 version. This module allows programming of turning machines with a C-axis milling head. A screen shot of this module is shown below.

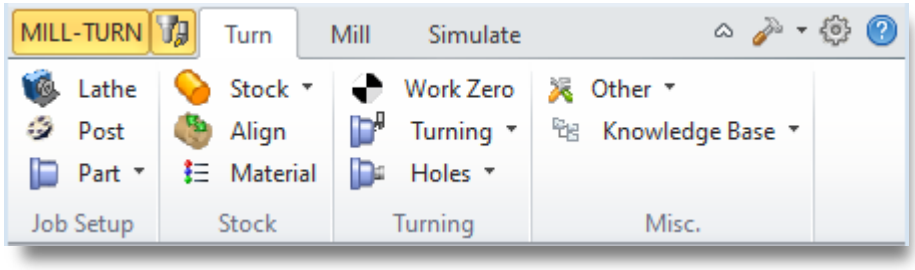


© 2020, MecSoft Corporation



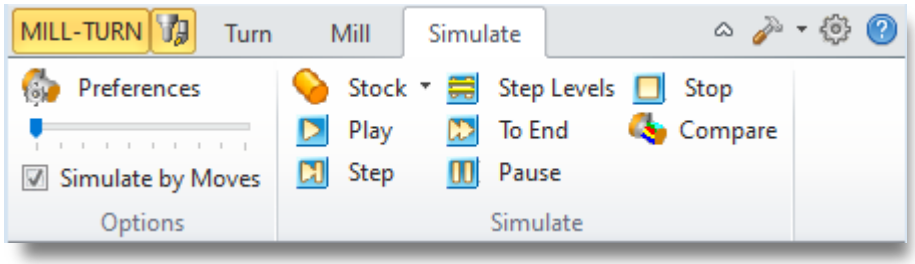
TURN TAB FUNCTIONS

The Turn tab is used to program turning operations on the part. Turning operations are created under a separate Setup icon in the Job tree of the Machining Operations browser.



MILL TAB FUNCTIONS

The Mill tab is used to program milling operations using the C-Axis head on the part. Milling operations are created under a separate Setup icon in the Job tree of the Machining Operations browser.

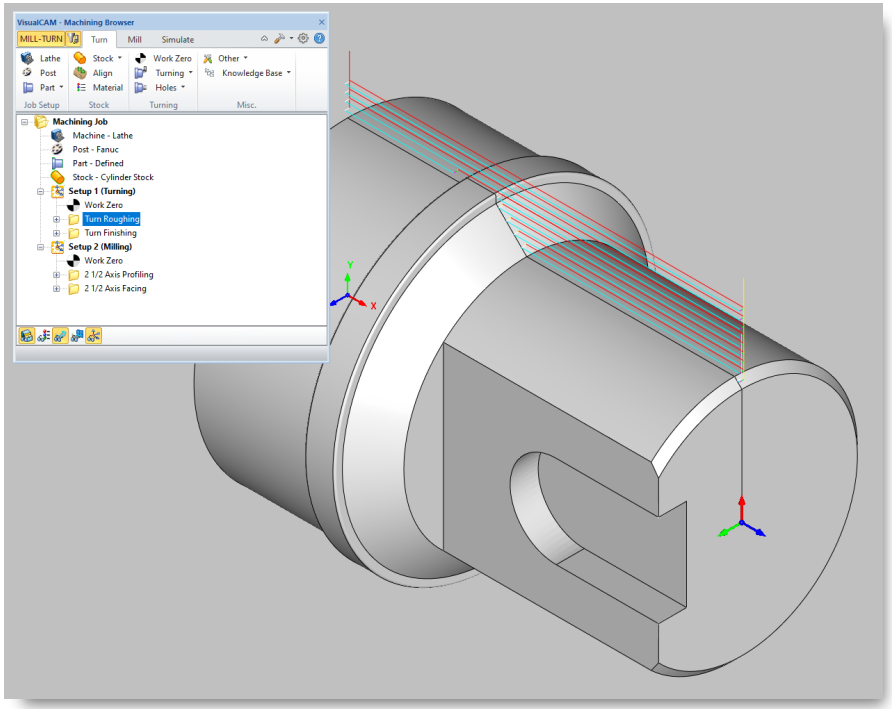


SIMULATE TAB FUNCTIONS

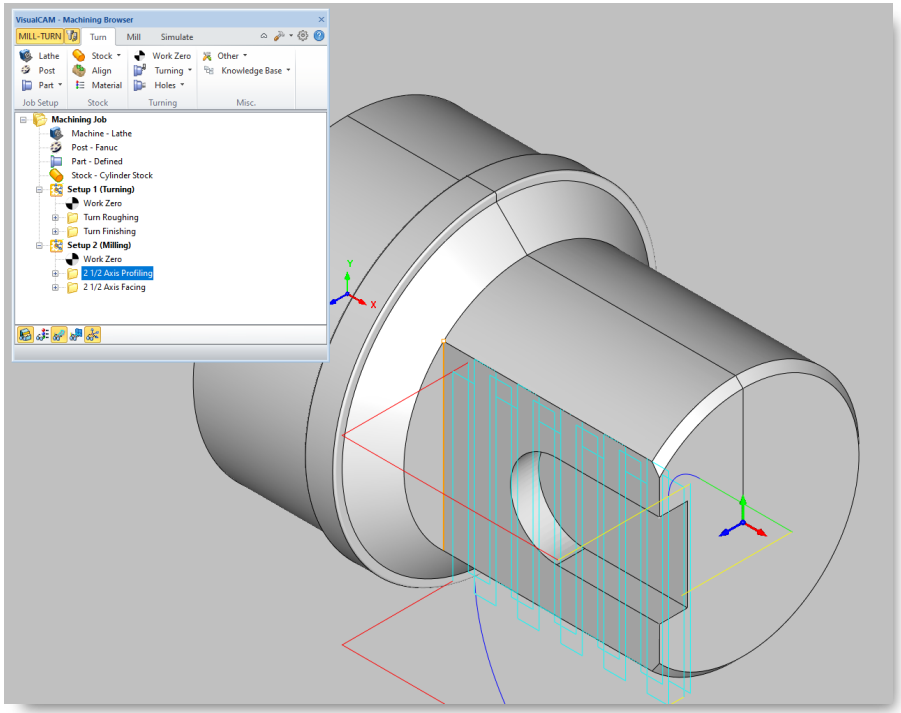
The Simulate tab is used to simulate the created operations, including both the Turn and Mill operations present in the file.

© 2020, MecSoft Corporation

Screen shots of Tuning and Milling operations performed in the same part using the MILL-TURN module is shown below:



Turn roughing operation



Mill Facing operation on the same part

© 2020, MecSoft Corporation

COMMON CAM ENHANCEMENTS

This section describes the common enhancements and changes to VisualCAM 2020.

1. A new version of the windowing system for VisualCAM has been integrated into the 2020 product. Additional window styles have been added to the CAM product as a consequence.
2. New simulation libraries from Machineworks Inc. has been incorporated into all the machining modules. These libraries have improved multi-threaded performance significantly as well as fixed many reported issues.

LICENSING ENHANCEMENTS

The following enhancements have been implemented in the licensing system for VisualCAM.

1. Cloud hosted concurrent network licenses using network authentication service has been implemented. This is a simpler scheme for implementing multiple licenses on school and other organizational networks. This scheme will work only when client machines have constant internet access.
2. The license administrator can now push license settings to the HKLM registry to be used by network users without having to enter them at each client machine.
3. Password fields are now protected and not displayed in the case of settings being pulled from the HKLM registry. This is important to large enterprises and educational institutions.
4. Maintenance expiry date is now being displayed in the license status dialog. Also a prompt when maintenance is close to expiry will be shown at the start of the product.

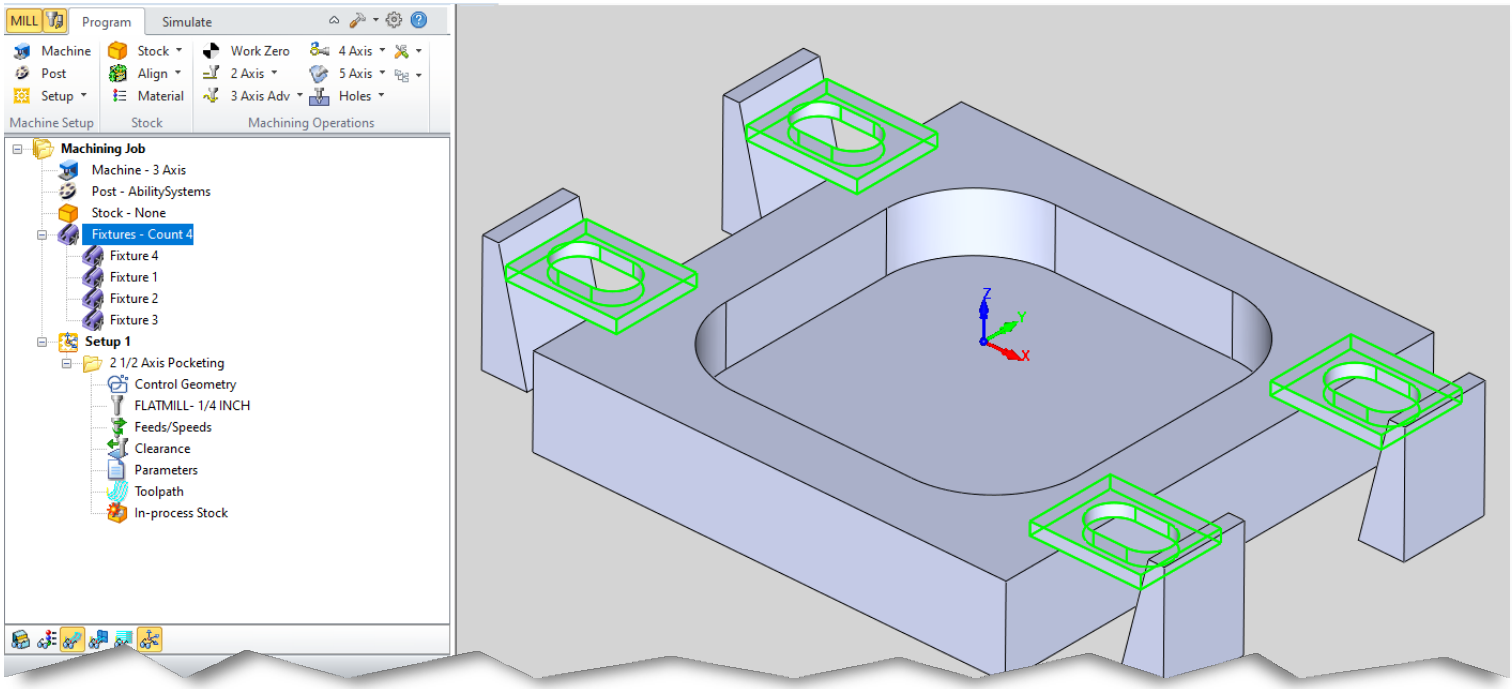
WHAT'S NEW IN VISUALMILL 2020

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

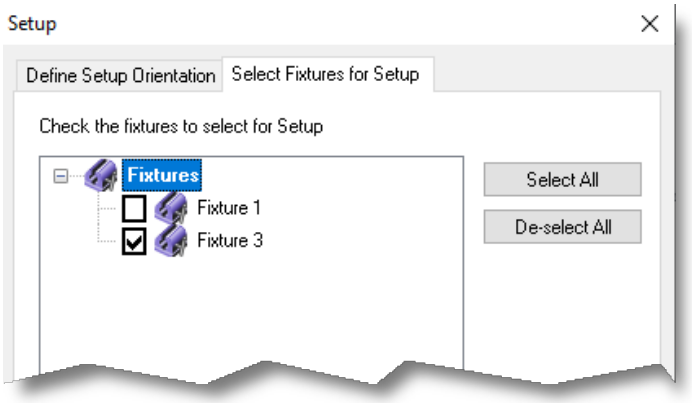
USABILITY ENHANCEMENTS

1. Fixtures can now be defined in the MILL module. Collision detection of generated toolpaths with fixture geometry has also been implemented. Global fixture geometry can be selected and are displayed on the Machining Operations tree in the browser dialog.

© 2020, MecSoft Corporation



Screen shot of the user of fixtures and their representation in the Machining Job tree in the browser window is shown above

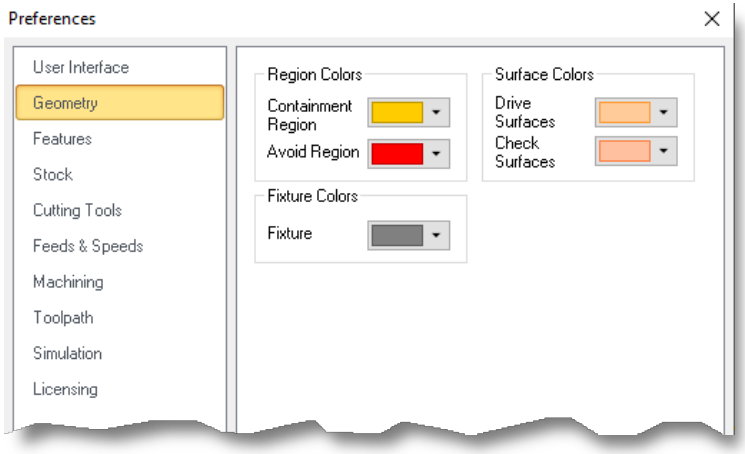


FIXTURES IN SETUPS

Once defined, the fixtures can then be added to any Setup to enable their use in the Setup's machining operations.

© 2020, MecSoft Corporation

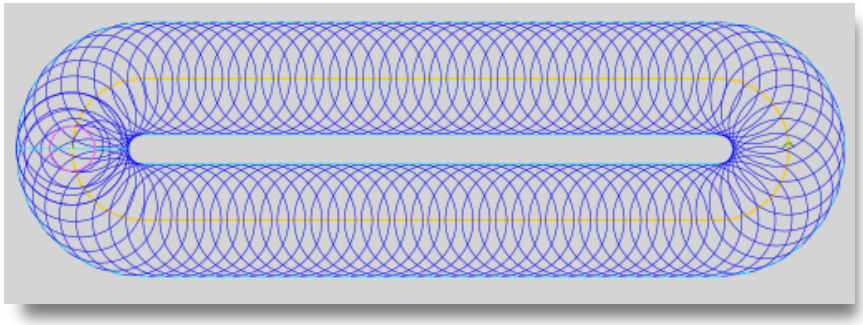
2. Preferences dialog has been reorganized based on object and function specific dialogs to stream-line preferences.



3. Better display support for high resolution (4K) has been implemented with newer scalable function icons in various toolbars and ribbon bars.

2-AXIS ENHANCEMENTS

1. A true trochoidal cutting path has been introduced.

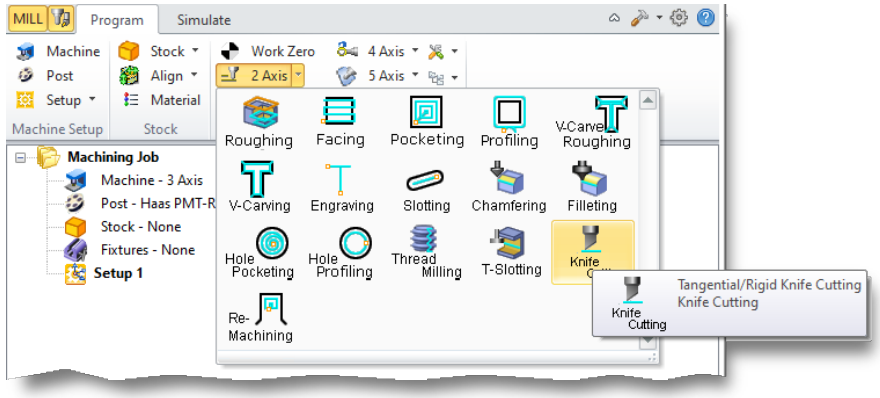


TRUE TROCHOIDAL MACHINING

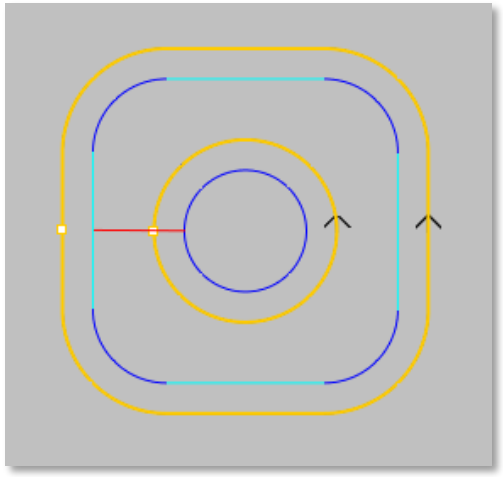
This option can be used to cut slots as well as other confined areas in hard material. The user selects an open or closed curve and specifies a trochoidal width to machine.

© 2020, MecSoft Corporation

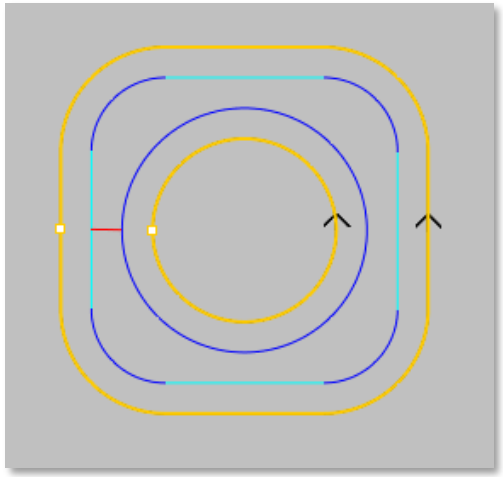
2. Programming with rigid knives can now be done. This option has been added to the existing knife cutting operations. The knife will be retracted and rotated into the new orientation when this option is checked.



3. Cut side in profiling can be set based on nesting of profiles. The system uses the user defined settings for the outer most profile and then alternates the cutting sides for any inner profiles, regardless of the direction of the profile curves.



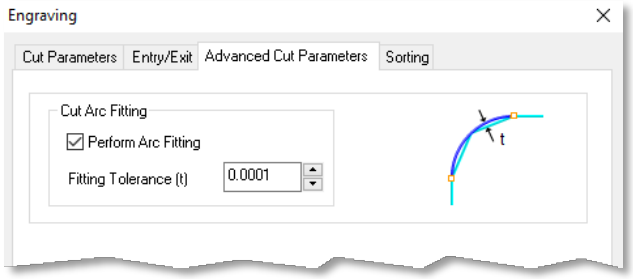
Profile toolpath with cut side on the left side of the curve



Profile toolpath with cut-side alternating based on profile nesting

© 2020, MecSoft Corporation

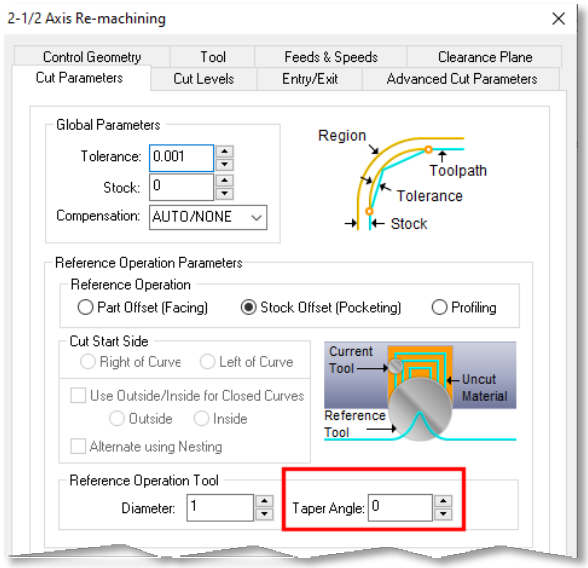
4. Profile machining start points have been enhanced in the case of toolpaths with corner extension.
5. The user now has an option to fit arcs to 2 ½ Axis Engraving toolpaths



ARC FITTING IN ENGRAVING

This option can be used fit arcs in the 2 ½ Axis engraving toolpaths.

6. Taper tools can now be defined as reference tools in 2 ½ Axis re-machining operations.



TAPER TOOL IN RE-MACHINING

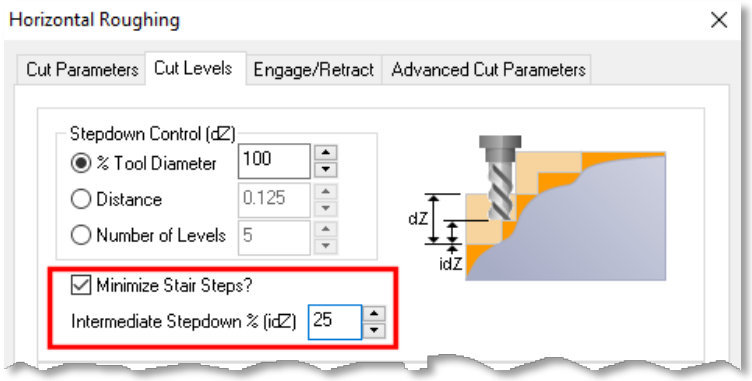
The taper angle can be set to create a taper or conical reference tool to create a 2 ½ axis re-machining toolpath

© 2020, MecSoft Corporation

7. Ramp height for ramping cut entries is set automatically now based on depth of cut in 2 ½ Axis machining modules. The tool now ramps from the previous cut level if the user sets a ramp height smaller than the height from the previous level to the current level.

3-AXIS ENHANCEMENTS

1. Step Minimization Roughing (SMR) option has been introduced in Horizontal Roughing. The Horizontal Roughing operations can leave a large stair-steps that need to be cleaned up by an additional operation – typically a re-roughing operation. A more efficient method will be to rough the part using the largest step-down the tooling, material and other cutting conditions will allow and then cut the stair-step left by this step *after* the lower cut level has been machined.



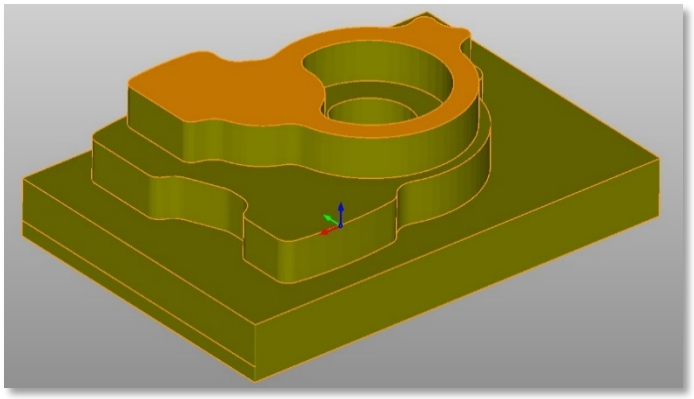
STEP MINIMIZATION ROUGHING

Specifying intermediate steps will create Z levels between the original cut levels that will remove the stairsteps left by these cut levels.

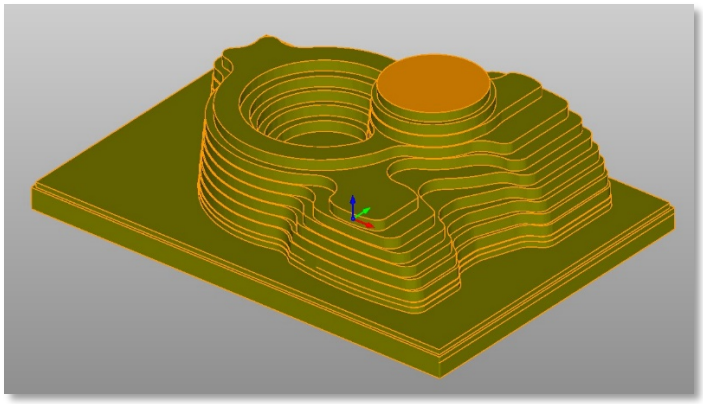
The advantages of this type of machining method are as follows:

a. Decreased time for machining
b. Lesser number of cutting tools used

© 2020, MecSoft Corporation



Part machined without Step Minimization Roughing. Here the maximum depth of cut allowed for tool has been applied.



Step Minimization Roughing applied with intermediate steps. This allows the significant reduction of stair steps.

2. Ramp height for ramping cut entries is set automatically now based on depth of cut in Horizontal Roughing and Horizontal Re-roughing modules. The tool now ramps from the previous cut level if the user sets a ramp height smaller than the distance to the previous level.

4 AXIS TOOLPATH ENHANCEMENTS

1. 4 Axis non-projection toolpath algorithms have been re-written and made more robust to handle more complex cases.

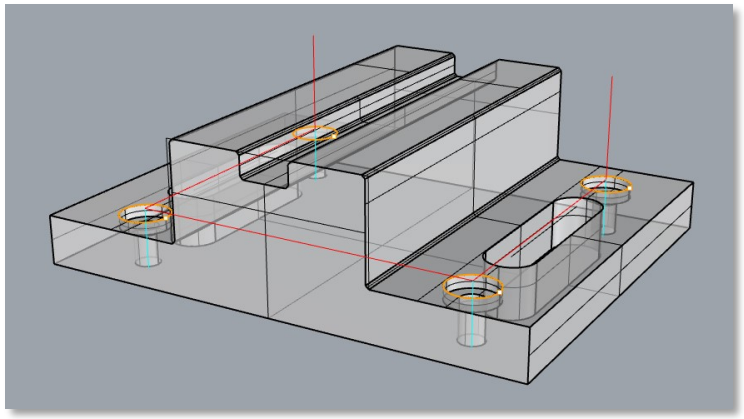
5 AXIS TOOLPATH ENHANCEMENTS

1. New Moduleworks toolpath generation libraries for 5 axis machining have been integrated with the product.

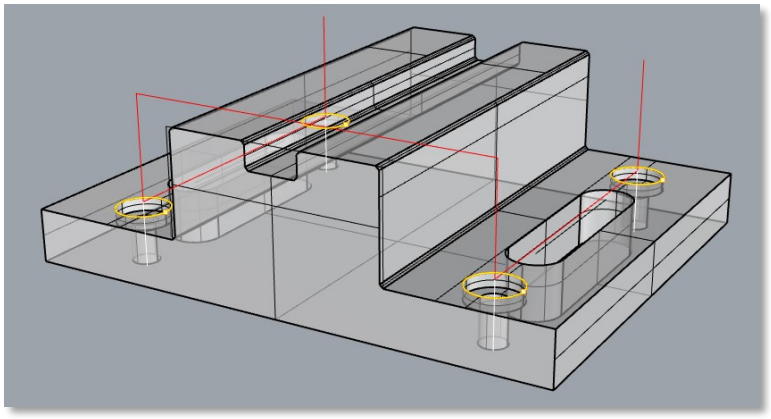
DRILLING ENHANCEMENTS

1. Skim clearance for drilling operations was implemented.

© 2020, MecSoft Corporation



Version 2019 behavior when Skim Clearance is set in drilling methods.

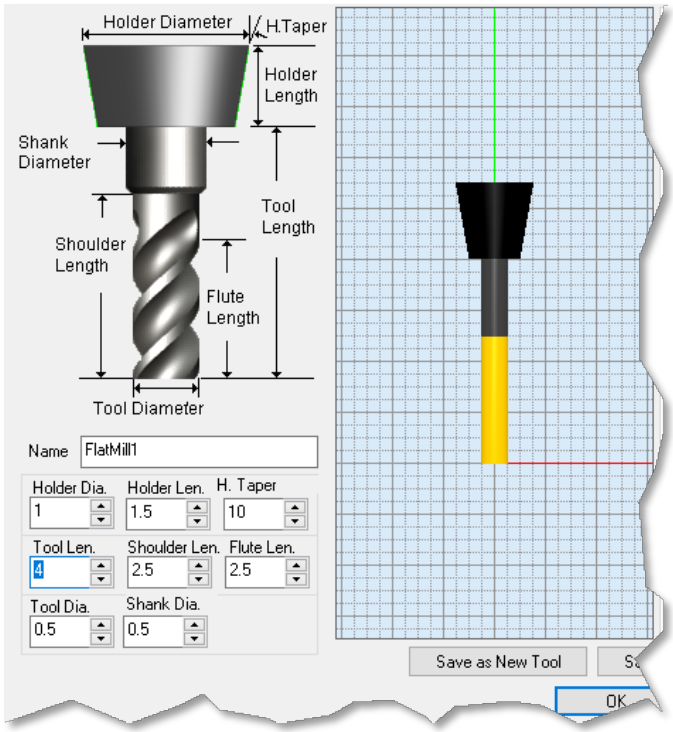


Version 2020 behavior when Skim Clearance is set in drilling methods.

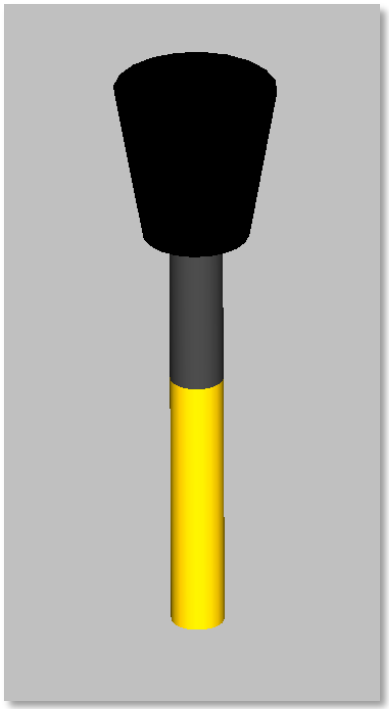
TOOLING ENHANCEMENTS

1. Taper tool holders can now be defined for all mill and drill tools. These taper tool holders will participate in collision detection and other functions involving the tool holder.

© 2020, MecSoft Corporation



All tool creation/edit dialogs have been updated to include taper holder definition.



Preview of a flat tool with a taper holder.

2. A new format tool library using CSV files has been implemented. CSV files now are segmented based on tool types.

FEEDS/SPEEDS ENHANCEMENTS

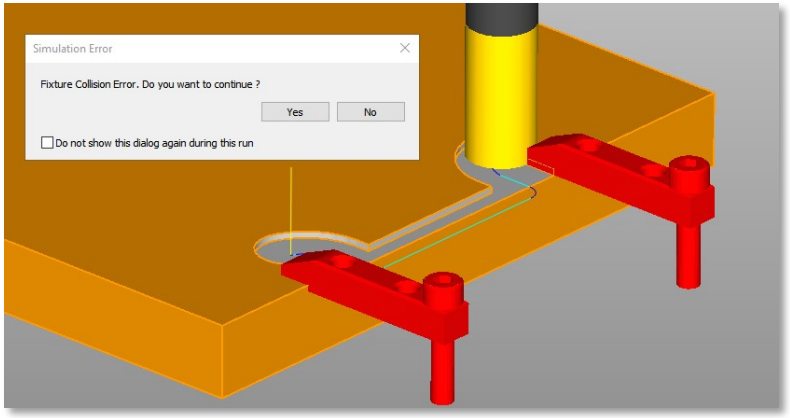
1. Feeds & Speeds information stared with the tool are now written out to the CSV format tool libraries.

© 2020, MecSoft Corporation

SIMULATION ENHANCEMENTS

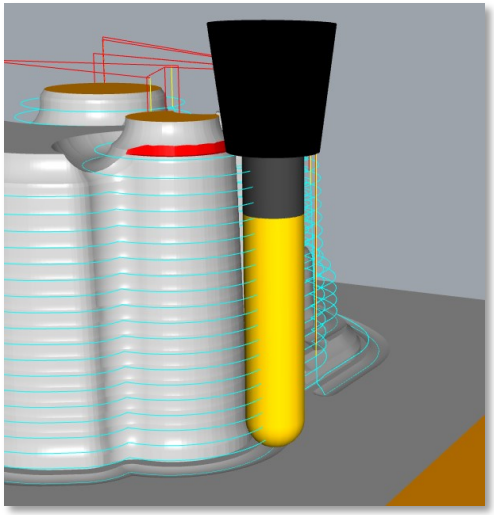
1. New simulation libraries were incorporated that fixes several issues as well as introduce performance improvements was implemented.

2. Collision detection between tool and fixture geometry was implemented. Detected collisions are displayed in red in the stock model.



© 2020, MecSoft Corporation

3. Collision detection between tapered tool holders and geometry was implemented. Detected collisions are displayed in red in the stock model.



MACHINE TOOL SIMULATION ENHANCEMENTS

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

KNOWLEDGE BASE ENHANCEMENTS

1. Selection rules for defining work zero automatically in Knowledge Bases has been implemented. This allows for parametric definition of Work Zeros when a Knowledge Base is loaded. As an example, the user can specify the Work Zero to be at the South-East corner of the stock model and save a Knowledge Base. When this Knowledge Base is loaded, the work zero will be automatically set to the South -East corner of the current stock model. Previously the work zero would be set to the saved coordinates.

POST PROCESSOR ENHANCEMENTS

1. Ability to modify the APT CLS file using a separate program, during output was implemented

© 2020, MecSoft Corporation

2. Rapid motion in all indexed operations are converted to use federate motions when setup is not aligned with the machine Z. This is done only when a head is defined and output to Local Coordinate system is NOT set.
3. Drill cycles need to be converted to simulated cycles if the setup is not aligned with the machine Z. This is done only when the machine has a head defined and output to Local Coordinate System Is NOT set.
4. New variables for MaxZ and MinZ output in the post have been added
5. If stock is not defined, UNDEFINED is output for the STOCK_X, STOCK_Y & STOCK_Z and other stock related variables
6. The issue with the Post-Processor Generator dialog dominating the display has been fixed.

BUGS FIXED

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

WHAT'S NEW IN VISUALTURN 2020

A new MILL-TURN module has been introduced. In addition to this, multiple customer reported bugs were fixed in the TURN module.

© 2020, MecSoft Corporation