



## What's NEW in VERICUT 8.0

IMPORTANT! - Licensing is NOT included in software shipments. See "How to get a license" below for details.

September 8, 2016

Dear VERICUT® User:

Thank you for your continued investment in VERICUT, an important part of your NC programming and machining process!

The VERICUT 8.0's NC program simulation, verification, and optimization technology is packed with new features making it more powerful and easier to use. This letter describes important changes in VERICUT 8.0. Take a moment to review what's new and improved in this release.

### Maintenance and Licensing Information

**How To Get a License** - All users must complete and return the License Request Form in the CD booklet, or submit the application at [http://www.cgtech.com/vericut\\_support/request-license/](http://www.cgtech.com/vericut_support/request-license/). Licensing is sent via Email only.

**NOTE:** This software requires a VERICUT 8.0 license.

*Software maintenance keeps you on the cutting edge* - CGTech provides update software to customers with current software maintenance. Your continued maintenance ensures that you have the most advanced verification technology available. If your maintenance has expired, please contact your CGTech representative (<http://www.cgtech.com/about/contact-us/>).

Sincerely,

Gene Granata

CGTech VERICUT Product Manager

## VERICUT 8.0 Release Notes

September 8, 2016

VERICUT 8.0 is available for 64 bit Windows, and is supported on Windows 7 and Windows 10 computers.

VERICUT's license server will continue to run and be supported on 32 bit or 64 bit computers.

Both VERICUT and the license server can be installed by both 32 bit and 64 bit computers. When installing VERICUT with a 32 bit computer you will be warned that it can only run on 64 bit systems. The warning will not display when installing the license server.

## V8.0 Enhancements

### New Ribbon Bar

VERICUT 8.0 optimizes workflow and helps you find the functionality you need with a minimum number of mouse clicks. Upon upgrading to VERICUT 8.0, the first thing you will notice is the new Ribbon Bar. The Ribbon Bar allows you to quickly select the VERICUT function you want to use. As you move from tab to tab, the Ribbon bar dynamically updates to show the options available for that tab. Options are grouped by the function they perform within VERICUT.

The Ribbon Bar is highly customizable, but to ease the transition, a "VERICUT Classic" setting organizes all of the menus and options where you previously found them (a "Quick Reference Guide" is also available on the first page of the Getting Started section of VERICUT and VERICUT Drill and Fastener Simulation Help or from the [cgttech.com](http://cgttech.com) website). Several other layout options are included and can be selected depending on the task at hand. You can also create and save your own ribbons as needed for different jobs.

### "Air Cut" optimization is built-in

You now have the ability to optimize "Air Cuts Only" (off-part milling cutter motions) with a new optimization feature included with VERICUT's base Verification license. The new "Air Cuts Only" optimization method is intended as an easy-to-use, entry level method of optimizing NC programs. Additional optimization strategies are available with the purchase of OptiPath or FORCE.

## **Read STEP files – no extra license required**

Read STEP files (.stp or .step) containing AP203 and AP214 (geometry only) protocols directly in VERICUT's modeling interface to describe machine, stock, fixture, and design shapes, or 3D cutting tool shapes in VERICUT's Tool Manager. The Tool Manager displays the CAD Geometry window that allows you to identify which parts of the CAD model file correspond with holders, cutters, or inserts.

## **Integration with cutting tool suppliers and tool management systems**

As with all software programs, the accuracy of the data input will directly affect the output. So, an accurate model of the cutting tool and holder is required for effective and accurate simulation of the machining process. Most leading cutting tool manufacturers now make 3D solid model data available and VERICUT can read in this model data for use in the simulation process. Many of the 3D models are available via the Machining Cloud App, and version 8.0 has been enhanced to take advantage of more Machining Cloud metadata. This can significantly simplify the configuration of tools for use in VERICUT, and better describe their proper use and limitations. VERICUT also integrates with most major tool management systems. Pre-setting suppliers including Zoller and Speroni can also interface to the software, so tool offsets and exact dimensions can be applied to the simulation session.

## **Tool Manager Preferences**

A new preferences dialog allows users to setup many default settings that help to streamline creation of cutting tool assemblies. For example, specify the colors used, the driven point offset numbering scheme (default to "1", or follow the tool number), CAD model tolerances for cutters and holders, and if a holder is desired (or not) for a new tool. Each imported 3D tool model can also have its own separate model tolerance. For example, you may want a very high accuracy on the cutter model, and less tolerance on the holder model. Version 8.0 Tool Manager has a number of other time-savers, such as being able to mark any tools as "Default", to be the starting point for building similar tools of that type, instead of starting from scratch. There are also a number of Copy/Paste/Multi Select improvements.

## **Consolidated Reporting Features**

The VERICUT logger now displays messages and reports from many sources in a tabbed logger window. In addition to errors, warnings, and other messages from the VERICUT session, it also displays information from AUTO-DIFF and X-Caliper. The messages can be grouped, sorted, and displayed in a variety of ways depending on user preference. VERICUT speeds investigation with dynamic links between logger messages and responsible NC Program records. The exact source of an error is just one click away.

## **Graphical Tool Path Analysis**

A new “Toolpath Trace” feature creates a wireframe of the motion path that can then be measured via VERICUT’s X-Caliper measuring tool. In NC Program Preview or Review modes, picking on a toolpath trace in the wireframe automatically sets the simulation to that record in the NC program.

## **FORCE Enhancements**

The FORCE module, uses physics-based optimization to determine the maximum reliable feedrate for a given cutting condition based on: force on the cutter, spindle power, maximum chip thickness, and maximum allowable feedrate. In version 8 there are new features for better control of entry/exit speeds, “clean-up” feedrates, and tooling information has been re-arranged to be more intuitive.

## **Modeling Enhancements**

Translate models via selecting solid model features, eliminating the need to create coordinate systems for positional information.

Enhanced modeling options provide greater control and flexibility over moving individual models, assemblies, and component origins, greatly reducing time to model machines, especially from 3D CAD model assemblies.

## **Other Enhancements**

VERICUT 8.0 has many performance enhancements.

A new method for comparing the stock cutting position to a 2D DXF file drawn to represent the design is added.

WinTool interface launches from within Tool Manager.

Windows 10 is supported.

Self-guided training sessions, launched from the Welcome Screen, automatically open the associated sample files.

## Enhancement Details

### **AUTO-DIFF report:**

- now displayed in the VERICUT Logger enabling you to select AUTO-DIFF deviation records while in NC Program Review.
- now displayed in the VERICUT Logger enabling you to select AUTO-DIFF deviation records and have the record that created the deviation highlighted in the NC Program panel.
- enhanced to automatically center on the location of an AUTO-DIFF Report deviation. When a deviation record is selected from the list in the AUTO-DIFF report Logger window, the Workpiece view or Profile view is centered on the deviation marker. Once you have selected the first deviation record, you can use the Up and Down arrow keys on the keyboard to cycle through the remaining deviation records. The graphic display will be updated for each deviation record.

### **CAM Interfaces**

#### **GibbsCAM-to-VERICUT Interface (GibbsV):**

- retrieve GibbsCAM intermediate tooling models and pass them to VERICUT.
- support Gang Tools.

#### **Mastercam-to-VERICUT Interface (MCAMV):**

- provide the ability to define a sub-register value when configuring Work Offsets.
- populate the component list with the Project Template by default unless a setup template is browsed for.
- pass Mastercam Thread Mill cutters to VERICUT.
- support Mastercam 2017.
- cutter compensation value is only output when the Mastercam Compensation type is "Control".

#### **NX-to-VERICUT Interface (NXV):**

- support the NX Core Drill subtype.

## FORCE

FORCE is enhanced with new features for better control of entry/exit speeds, “clean-up” feedrates, and tooling information has been re-arranged to be more intuitive.

New checks are added to report when a Force optimized NC program exceeds the specified Force Limits.

FORCE now checks during the optimization for when optimized chip thickness exceeds the specified Force limit due to the Min Cut Feedrate being too high. The following warning is output:

“Warning: Chip Thickness limit for tool *"id"* exceeded at record number *"record number"*. Optimized feedrate limited by Minimum Cut Feedrate value *"value"*.”

The same check is also done for Force or Power limits if the Force optimization is by "Force and Chip Thickness" or "Power and Chip Thickness", respectively. The corresponding warning messages are:

“Force limit for tool *"id"* exceeded at record number *"record number"*. Optimized feedrate limited by Minimum Cut Feedrate value *"value"*.”

“Power limit for tool *"id"* exceeded at record number *"record number"*. Optimized feedrate limited by Minimum Cut Feedrate value *"value"*.”

New checks are added to report when a value exceeds the specified ranges for Chip Thickness, Surface Speed and Rake Angle.

All 3 values (chip thickness, surface speed, and rake angle) are checked on each motion during the simulation, but the corresponding warnings are only output the first occurrence of the out-of-range condition per tool. The following warning messages are output:

### 1. Chip Thickness:

Chip Thickness Only optimization:

“Warning: Tool Force Chip Thickness *"value"* mm is beyond Force Material Chip Thickness range for tool *"id"*. Reduce Chip Thickness Limit.”

Force and Chip Thickness optimization:

“Warning: Tool Force Chip Thickness *"value"* mm is beyond Force Material Chip Thickness range for tool *"id"*. Reduce Chip Thickness/Force Limits.”

"Power and Chip Thickness" optimization:

“Warning: Tool Force Chip Thickness *"value"* mm is beyond Force Material Chip Thickness range for tool *"id"*. Reduce Chip Thickness/Power Limits.”

2. Surface Speed:

'Warning: Tool Surface Speed "*value*" m/min is beyond Force Material Surface Speed range for tool "*id*".'

3. Rake Angle:

"Warning: Tool Force Rake Angle "*value*" (rad) is beyond Force Material Rake Angle range for tool "*id*"."

Note that for Chip Thickness and Surface Speed only the maximum values are checked while for Rake Angle both the maximum and minimum values are checked. The reason is that for ball-nose cutters, the areas on the tool that are very close to the bottom might produce very small values of chip thickness and surface speed which are well out of material calibration range but such areas make a negligible contribution to the total force evaluation and can be safely ignored.

## G-Code Processing

New macro, **CycleMillPocketCircleCuttingRapid**, is added to support G12, G13 Rapid traverse setting in Tosnuc 888 controls.

New macros **CircleMotionPlaneZX**, **CircleMotionPlaneXY**, **CircleMotionPlaneYZ** are added to support cases using vertical circles when the **CutterCompFull** macro is set with an Override Value = 1 (full look-ahead mode). These macros differ from the **MotionPlaneZX** macro in that the new macros only apply to the current circle and do not change the motion plane.

New macro, **CutterCompRapidSuspend**, enables you to suspend Cutter Compensation for rapid motions.

VERICUT is enhanced to support Heidenhain continuation characters of the type "CYCL DEF 200 DRILLING Q200=+2 Q201=-60 Q206=+150 Q202=+5 Q210=+0 Q203=+0 Q204=+50 Q211=+0" by defining it in the Word Format table using the new Type=Special and Sub Type=Continuation and using the SetPreProcessorType macro with an Override Value=6.

The Toshiba (Tosnuc) control type is enhanced to support an 'O' word immediately followed by an alphanumeric subroutine name.

The following error message "Error: Invalid type of argument passed to function "sqrt"" is now output when a negative number is passed to a square root function.

Support for Okuma OSP Local Variables PA-PZ is enhanced to allow local variables beginning with P to be referenced without being previously defined, and removes the requirement that it is followed by an "=". Since all local variables are not being defined with the "CALL" statement, to avoid finding a local variable defined in a parent sub, you

should call the **SetLocalVarAccessType** macro with an Override Value=1 during start of processing.

The processing of a CGTECH\_MACRO statement is enhanced to support tab separated arguments in addition to space separated arguments. While this routine has been enhanced to support tabs, it is generally not recommended to use tab separated arguments.

New macro, **OptiSetMaxIntervals**, is added to enable you to override the limit for the number of OptiPath subintervals. The default limit is 512.

New macro, **ApplyCircleMappingToTransform**, is added to enable you to specify whether or not to the mapping associated with the **CirclePresentAxis** macro should be used with transformations.

Unexpected Z-axis motion no longer occurs when applying the gage offset with Siemens 840D TRANSMIT Polar Interpolation active.

In a specific multi-channel (sync) project file, all channels are not synced do to a motion block with Feedrate set to 0, creating motion with time "infinity", which never gets processed. A check is added for a Feedrate value of zero while in CSS mode. An error message is output and the time for the current line is set to 1.

The **SpindleOrient** macro restores the active spindle if there was no motion on the block.

New macro, **CycleTurnStockY**, is added to enable you to specify an additional offset in the Y direction for clean-up stock machined by the roughing cycle after the roughing passes. It can also be used to define the total stock being removed by a **CycleTurnRoughLoop** macro cycle.

New macro, **CycleTurnGrooveStartIncre** is added to define the shift for the Okuma G73 "I" value and the shift for the Okuma G74 "K" value.

New macro, **AutosetTableAxisVars2**, defines a table for bi-directional variables such as Renishaw cycles which read/write to the #2500 system variables.

Macro, **NumSubSequenceLoop**, has been completely re-written.

## Machine Simulation

The check for interpolated motion has been enhanced, so that it only looks at the axes that have been directed to move, and the axes that are being mapped to XYZ. Axes that are not related to motion are not considered when processing RPCP.

Machine Simulation is enhanced so that when Stop At=Collision is active, if both a near-miss and a collision occur, the simulation will first stop at near-miss point and then at the collision, and both near-miss and collision errors will be reported.

## **OptiPath**

A new "Air Cuts Only" optimization method is added to the Optimize Control window. This feature enables you to optimize all milling cutters for air cuts without having to create OptiPath records for all of the tools individually. This feature is included with VERICUT's base Verification license.

## **Reviewer**

Reviewer is enhanced so that the Gage Point value is now displayed in the Tools panel (Home tab > Analysis group > Tools).

## **Tool Manager**

The Turning Groove Insert definition is enhanced to not allow the value of the D parameter to be less than or equal to the value of the R parameter. If "D" value is less than or equal to the "R" value, an error is output and the insert is not created.

Cutter Compensation values, that can be defined in Tool Manager, now accept ID's entered as 01, 02, 03 etc.

Import CAD Tool is enhanced to read an NX part file to create a tool.

Tool Manager is enhanced to enable loading an "Optional Tool Assignment List" from a spread sheet.

## **Verification**

VERICUT is enhanced to display the actual RPM in the Status panel when using Constant Surface Speed.

The File Summary window "Copy selected files to" option (Directory or Zip File) is now saved in the Preferences file.

The Okuma OSP G-Code Log, Debug Variables is enhanced to show local variables being defined on a block in addition to the referenced variables.

The VERICUT logger is enhanced to enable you to Sort Errors by Tool while in NC Program Review.

The VERICUT Logger is enhanced to color code different types of messages. If the Use Color feature in the right mouse button menu is active:

- Error messages are displayed in red text.

- Warning messages are displayed in blue text.

- Information messages are displayed in black text.

Over-travel error messages in Logger now provide the line number where the error occurred.

The VERICUT Logger is enhanced to enable you to sort entries by Message Type, by Tool and by NC program.

VERICUT is enhanced so that when a collision occurs between a tool's Stem/Shank and the Cut Stock, it is now reported as a Stem/Shank collision (not a "Holder" collision).

### **VERICUT Drill and Fastener (VDAF) Simulation**

Collision errors are now highlighted in the VDAF Logger and can be sorted by Error Message Type.

In VDAF, options Set Spin Center, Snap to Orthogonal View and Active Coordinate System have been added back to the "View" right mouse button menu.

VDAF's VcPost file is added to the VDAF File Summary window.

VDAF collision reporting is enhanced so that when Stop At=Collision is active, if both a near-miss and a collision occur, the simulation will first stop at near-miss point and then at the collision, and both near-miss and collision errors will be reported.

Access to Fastener Programming added to the Project tab > Fastener group.

Access to Simulation Location Report and Fastener Summary are added to the Info tab > Fastener Group.

### **VERICUT Drill and Fastener Programming**

VDAF Programming is enhanced to enable adding a CSYS location to be manually added in a sequence.

VDAF Programming is enhanced to enable graphically orienting the I and J axes of a CSYS to fix the final position of the machine.

Access to Fastener Programming is added to the Project tab > Fastener group.

### **Miscellaneous**

The VERICUT installer is added to optionally install stand-alone Tool Manager.

VERICUT is enhanced so that when a VERICUT session starts, it checks if there are orphan ~vcreviewer\_setupxxx files in the temporary folder and deletes them.

Mastercam and SolidWorks are added to the Dynamic Controls feature.

The Progress Bar color has been enhanced for all Color Themes to provide better visibility.

VERICUT is enhanced so that if you have the VERICUT window "maximized" when you exit the VERICUT session, it is saved in the Preferences file, so that the VERICUT window will be "maximized" when you start a new VERICUT session.

The feature that controls Coordinate System visibility is now consistent throughout VERICUT.

In the Settings window: Properties tab, the "Replace Material When Stepping Back" label has changed to "Replace Material During Review". The "Animate Machine When Stepping Back" label has changed to "Animate Machine During Review".

The X-Caliper Feature/History feature is enhanced to highlight the block in the NC Program panel that created the selected feature.

A new option, **Project Units**, is added to the Control Settings window (Machine/Control tab > Control Group > Control Settings command button): Motion tab, Default Control Units feature, enabling you to specify to use the project file units in addition to Inch and Metric.

A new option, **Ignore Direction**, is added to X-Caliper Highlight Same Plane/Cylinder that enables you to find all parallel planes regardless of the surface normal orientation.

## Problems Resolved in V8.0

### AUTO-DIFF

AUTO-DIFF results for a specific project file where the design profile in many places coincides with the cut stock profile is now consistent.

### CAM Interfaces

#### CATIA V6-to VERICUT Interface (CATV6):

- correctly passes tool units and tool numbering for tools imported from CATIA V6
- no longer takes an extremely long time to pass a specific 3DXML file imported from CATIA V6 to VERICUT. The time required to load is reduced from 45 minutes to a few seconds.
- CATIA V6 Stock and Design models to VERICUT in the correct position.
- passes CATIA V6 Fixture models (Clamps) to VERICUT in the correct position.

#### GibbsCAM-to-VERICUT Interface (GibbsV):

- no longer changes the color of the fixture component.
- transfers a specific model from GibbsCAM to the correct position in VERICUT.
- transfers all tools in a specific GibbsCAM file to the VERICUT Tool Manager.
- transfers a specific GibbsCAM turning tool with 2 driven points to the VERICUT Tool Manager.
- correctly transfers Back Bore and Bore tools in a GibbsCAM file to the VERICUT Tool Manager.
- transfers 2 tools in a turret from GibbsCAM to the correct turret positions and orientations in VERICUT.
- correctly transfers intermediate tooling in a GibbsCAM file to VERICUT.
- correctly transfers a specific inserted tool with the correct driven point from a GibbsCAM file to VERICUT.
- GibbsV for GibbsCAM 2014 now correctly writes out to the gibbsv\_2014\_user.prefs file.
- GibbsV for GibbsCAM 2015 now correctly writes out to the gibbsv\_2015\_user.prefs file.

- GibbsV for GibbsCAM 2014 no longer locks up the GibbsCAM session if you shut down the GibbsV Interface with the GibbsCAM Post dialog box displayed.
- GibbsV for GibbsCAM2015, no longer closes the GibbsCAM Post dialog box and locks up the GibbsCAM session if you shut down the GibbsV interface with the GibbsCAM Post dialog box displayed.

#### **Mastercam-to-VERICUT Interface (MCAMV):**

- MCAMV for Mastercam X7 transfers a specific Mastercam X7 tool to VERICUT as a correct cutter profile.
- transfers a specific Mastercam Thread Mill tool to VERICUT with the correct thread pitch.
- The G-Code Offset now works correctly with MCAMV in Portuguese.

#### **NX-to-VERICUT Interface (NXV):**

- correctly passes a specific cutter and shank of a thread milling tool from NX10 to VERICUT.
- localized Czech language version, no longer outputs UTF8 data warnings.
- unexpected NX10 termination no longer occurs when clicking on the Options button in the NX-to-VERICUT Interface when using a specific NX part file.
- correctly passes the proper Flute Length values for center drill tools from Siemens NX10.

### **G-Code Processing**

The Okuma OSP Variable Tag VS[PJ] now creates a Global variable.

An error is now correctly output for situations where the feedrate exceeds the "Fast Feed (FPR)" when cutting circle records while in CSS mode. Previously, "Fast Feed (FPR)" for circle records while in CSS mode was not supported.

In a specific project file where the B axis is linear, using a B=DC(...) command now produces the correct B axis motion.

The **Siemens\_P\_PFRAME\_FI** macro with Override Text=ADD now produces the correct results.

The **MatchParensInComment** macro, using an Override Value=1, is enhanced to search for a matching "paired" (START COMMENT/END COMMENT) character rather than just looking for a matching "END COMMENT" character. The following START COMMENT/END COMMENT pairs are supported: (), [], {}, <>.

In a specific project file, engraving with CYCLE60 on a cylindrical surface with TRACYL and ATRANS and Cylindrical Interpolation active, now produces the correct results.

The unit conversion with a \$P\_UIFR (Siemens Frames) rotary value, in a specific project file, is now correct.

The first NC program file is no longer ignored, after using the MDI window, when the project has multiple NC program files. A false, "Already at end of file. Reset or rewind." message is no longer output for a project file that only has a single NC program immediately after returning from the MDI window.

Processing a specific project file using the **CutterCompFull** macro with Override Value=1, now correctly outputs an Error/Warning for a motion block that cannot be reached when the contour is compensated.

Processing a G34 threading cycle, in a specific project file, now turns the thread correctly.

The **TableSubRegisterValue** macro no longer changes the way that dynamic work offset works.

The Siemens sin840D MIRROR command used in combination with AROT and ATRANS now keeps the coordinate system oriented correctly.

The positions of two turning inserts, in the Machine/Cut Stock view and Workpiece view, in a specific multi-channel (sync) project file are now consistent.

In a control file, extracting the value of # \$P\_ACTFRAME[X,TR] with the **UpdateVariable** macro with Override Text = CGT\_SAVE\_ACTFRAME\_X\_TR;# \$P\_ACTFRAME[X,TR] now works correctly as it did in V7.3.1.

Unexpected Z-axis motion no longer occurs when applying the gage offset with Siemens 840D TRANSMIT Polar Interpolation active.

In a specific multi-channel (sync) project file, all channels are not synced do to a motion block with Feedrate set to 0, creating motion with time "infinity", which never gets processed. A check is added for a Feedrate value of zero while in CSS mode. An error message is output and the time for the current line is set to 1.

The **SpindleOrient** macro now correctly restores the active spindle if there was no motion on the block.

A custom drilling cycle in a specific project file, no longer keeps calling itself until the maximum subroutine level is exceeded, now works correctly as it did in V7.4.

In a specific project file, using the **CutterCompFull** macro with an Override Value=1 no longer produces the wrong Cutter Compensation motion for motions that only have X or Y movement. In addition, a false error on the Cutter Compensation cancel code is no longer output.

The **SiemensBasicOriginFrame** macro now correctly restores the Work Offset setting.

Unexpected VERICUT termination no longer occurs when processing a **SiemensRotaryAxisOrder** macro used in a specific project file.

The **CycleTurnRoughIgnoreFinish** macro now works correctly when used in a specific project file.

The "GON" function in a specific Z32 control file now works correctly.

An Okuma OPS G87 Turn Cycle no longer gets caught in a loop if it is used more than one time.

## Machine Simulation

Unexpected VERICUT termination no longer occurs when processing a C11.556 move in a specific project file. The **SiemensCompToVcAxisMapping2** macro is enhanced to support multi-digit commands such as C11.556.

Unexpected VERICUT termination no longer occurs when adding a Siemens CYCLE119 to a specific configuration.

Material removal in a specific project file is now correct.

When processing a specific project file, a false "Warning: Tricept singularity occurred!" is no longer output.

When moving from EIA (C=DC) back to linear, VERICUT now produces the same results that are produced on the actual machine.

The Driven Point is moved correctly with the tool tip for a Starrag ZT800 machine when processing a Siemens 840d TRAFOOF command.

When an SOR profile cannot be created from an insert due to the insert's orientation, VERICUT now outputs the following error message:

"Could not create SOR cutter from insert of tool "tool\_id". Material removal is disabled for this tool."

In the conditions where a milling tool has a badly oriented insert, VERICUT outputs the above error message, material removal is disabled, the cutter is not displayed in Workpiece views, holders are displayed, and Holder/Stock collision checking continues as usual.

## OptiPath

OptiPath no longer outputs two identical blocks to the optimized file when a block contains a G40.

When processing a specific NC program file with OptiPath, all optimized blocks are now output.

In Interactive OptiPath, clicking Play, then Play again now stops the simulation. Clicking Play repeatedly now toggles between Start the simulation/Stop the simulation.

## **Reviewer**

Reviewer now correctly displays all tools in the lower turret for multi-channel machines.

Reviewer now correctly displays synchronized tool motion in the Workpiece view on a multi-spindle/multi-tool machine.

Reviewer now correctly displays the stock transfer from the main spindle to the secondary spindle in a specific VERICUT reviewer file.

Reviewer now correctly opens a reviewer file when created from a project file using a tool library file that has an Ampersand symbol (&) in the file name.

Reviewer files created from a specific multi-setup project file that uses STEP models now run correctly in Reviewer.

Reviewer no longer displays the stock as moving when it should not be moving in a specific reviewer file.

## **Tool Manager**

Material removal for a specific project file, that is using FastMill with a concave cutter with a holder inside and the holders in the tool assembly are not ordered from lowest to highest in Z, is now correct.

Selecting a driven point for a tool now correctly highlights the selected driven point in the graphics area. A yellow marker indicates the selected driven point and a blue marker indicates that there is a duplicate point at the same location.

Dynamic Controls can now be set in the stand-alone Tool Manager Preferences window. The settings are stored in the cgtech\_toolman\_80\_user.prefs file.

## **VERICUT Drill and Fastener Programming**

The Sequence: Sort Column feature in VERICUT Drill and Fastener Programming now sorts the column correctly.

In VERICUT Drill and Fastener Programming, the CYCLE Parameters on the Fastener Programming panel: Post processor options tab now update correctly when switching between locations that have the same CYCLE assigned.

## Verification

When a referenced tool library is saved using File Summary, the path to the original tool library is no longer saved.

An invalid Collision error is no longer reported in a specific project file using imported 3D tool inserts.

The Tap tool trace in a specific project file is now displayed correctly when the Stock is spinning.

A specific project file using a CAD tool with 2 inserts now removes material in the correct direction.

A specific project file now runs without errors as it did in V7.1.4.

A false Constant Gouge Check error is no longer output when processing a specific project file.

Unexpected VERICUT termination no longer occurs while processing a specific project file using X-Caliper Air Distance after multiple Broach passes.

Unexpected VERICUT termination no longer occurs after using the View Cube Capture Layout feature after saving an In-Process file.

While processing a specific project file, the CYCLE95 no longer adds material in the Profile view.

Polishing tools now work correctly on turning stock. It does not remove material but "paints" over the stock where it touches it. Note that the "painting" on turning stock occurs only in Workpiece view. The Profile and Machine/Cut Stock views are not changed because polisher does not change the stock turning profile. When stock spindle stops, the stock display in Machine/Cut Stock view does show the "painted" areas.

In a specific project file, 3D Cutter Compensation now processes correctly.

Unexpected VERICUT termination no longer occurs, while processing a specific project file using a 3 channel Index C200 mill-turn machine, while all 3 subsystems are in feed motion (linear or circular).

In a specific project file, cuts are no longer suppressed while cutter compensation is active and the **CutterCompFull** macro is set to full look-ahead mode.

Unexpected VERICUT termination no longer occurs when processing a Sin840d Turn command when the **CutterCompFull** macro is set with an Override Value=1.

The visibility setting is now correct when a model is brought into the Project Tree using Drag & Drop.

Siemens Sin840d CYCLE800 command now sets the work offset at the correct position in a specific project file.

The Cutting Resolution graph on the Settings window: Properties tab no longer fails on computers that have 64GB of RAM.

Running the simulation of a specific project file now produces the same results in the Workpiece view and Profile view.

The interactive OptiPath window no longer restricts the re-sizing of the NC program window when both windows are displayed.

The Visibility status icons, of models in the Project Tree, now remain correct after changing the Color Theme.

False Insert/Stock collisions when the tool retracts are no longer output for a specific project file.

Changing the Cutting Resolution, either by typing a value in the Cutting Resolution text field or by dragging the icon in the graph, in the Settings window: Properties tab > Tolerance section no longer requires clicking on the OK button twice to accept the changes and close Settings window.

You can now Copy an NC subroutine from one Setup and Paste it to another Setup.

When re-sizing file selection windows, you can no longer drag the left hand border beyond the right hand border resulting in a zero width (invisible) window.

## Miscellaneous

The CAD Model interface is enhanced to support Pro/E part and assembly files with or without the revision number in the file extension. A specific Pro/E part file with a .prt extension is now correctly read into VERICUT using Spatial R26 SP1.

A specific project file using encrypted files and a VERICUT Single Platform license, now correctly reads In-Process files created with the In Process, AutoSave: with the File End set to the End of each File feature.

OptiPath and Force now support the program structure required when using Makino sub-programs.

NC Program Review no longer incorrectly displays the tool path, in a specific project file, when Set Start is set at the first line of program.

NC Program Review now correctly displays a specific tool, used in a specific project file.

The Tool Tip resulting from hovering over a word in the NC Program window no longer displays false "command word should not be in an expression" messages.

Batch Wizard scripts using the "batchp" command now correctly displays errors and messages in the Logger when VERICUT launches at the end of processing.

In NC Program Review, clicking Play, then Play again now stops the simulation. Clicking Play repeatedly now toggles between Start the simulation/Stop the simulation.

The **GlobalVariables** macro documentation has been updated to indicate that this macro needs to be called in the **Reset** event.

## **New Macros in V8.0**

**AbsoluteShiftLocal**  
**ApplyCircleMappingToTransform**  
**AutosetTableAxisVars2**  
**CutterCompConnectionType**  
**CutterCompLookAhead**  
**CutterCompOffAtTangentialExit**  
**CutterComRapidSuspend**  
**CycleMillPocketCircleCuttingRapid**  
**CycleTurnGrooveStartIncre**  
**CycleTurnStockY**  
**DisplayPin**  
**ljk2AnglesAddOffset**  
**ljk2AnglesFromComponent**  
**ljk2AnglesInput**  
**ljk2AnglesLinearIgnoreLimits**  
**ljk2AnglesLogic**  
**ljk2AnglesPickSolution**  
**ljk2AnglesRotaries**  
**ljk2AnglesToComponent**  
**ljk2AnglesVerticalRule**  
**ljkAAxisMotion**  
**ljkBAxisMotion**  
**ljkCAxisMotion**  
**OptiClose**  
**OptiSetMaxIntervals**  
**PolarAxisOrientationOnOff**  
**RotaryDirShortestDistLinear**  
**RotaryTypeLinearOption**

**SetBrotjeABComponents**  
**SetBrotjeZWComponents**  
**SetProjectInfoVars**  
**SiemensEXECUTEPullout**  
**SiemensParotAngles**  
**SiemensRotaryAngleConversion**  
**SyncCodeAlpha**  
**SyncIdsBinaryAlpha**  
**SyncNoWaitOption**  
**SyncWaitOnOff**  
**TapeMoveRollers**  
**TapeRebuildForm**  
**ToolInsertTurningPlane**  
**WPAbsoluteShiftLocal**  
**WPAbsoluteShiftLocalTable**  
**WPAAdjustOffsets**  
**WPCreate**  
**WPIcrementalShiftLocal**  
**WPIInput**  
**WPLocalOnOff**  
**WPMatrix**  
**WPMoveOnOff**