



What's NEW in VERICUT 7.3

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

November 13, 2013

Dear VERICUT® User:

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

The VERICUT 7.3'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 7.3. 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/. Licensing is sent via Email only.

NOTE: This software requires a VERICUT 7.3 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,

Bill Hasenjaeger

CGTech Product Marketing

VERICUT 7.3 Release Notes

November 13, 2013

VERICUT 7.3 Enhancements

Ease of Use Enhancements

New icons used throughout VERICUT and are available in three user selectable sizes. The new icons provide a more consistent design and better resolution.

Project Tree

The Favorites window enables you quickly access frequently used directories and files without using a file selection window each time. The Favorites window is completely configurable so that it can be set up to be the most beneficial for your work process. The Favorites window is stored in your user preferences file so that it remains available when you change project files.

The Side Tool Bar can be hidden and re-displayed, for provide easier Project Tree navigation.

The Configure menus are now all displayed a consistent size, but are resizable by the user for better Project Tree navigation.

New feature, **Model File Edge Display**, on the File menu > Preferences window: Startup tab turns On/Off model edge display in a Machine or Machine/Cut Stock view for better visualization.

New feature, **NC Program Editor**, on the File menu > Preferences window: PDF/HTML/Editor tab enables you to specify the */path/filename* of the NC program editor that you want to use. The setting is stored in the Preferences File. This feature replaces the environment variable CGTECH_TPEDITOR

A new **Start At / Stop At Options** window enables you to specify multiple Stop At (Breakpoint) conditions for easier simulation control.

G-Code debugging windows (Machine Offsets, Call Stack, and Subsystem Motion) are combined into a single tabbed window to provide easier navigation. The new G-Code Debug window is dockable and provides access to the Auto Hide feature.

New feature, **Snap to Orthogonal View**, is added to make it easier to get to a known orientation. Orient the view so that it is close to the orthogonal view that you want and then click on the **Snap to Orthogonal View** button and the view will re-orient to the exact orthogonal view. **Snap to Orthogonal View** can be accessed in the View Orient window, from an icon in the toolbar, and from the right mouse button menu in a view.

Enhancement Details

CAM Interfaces

The **CATIA-to-VERICUT Interface (CATV5)** is enhanced to support “Mirror” for Stock and Design models.

The **CATIA-to-VERICUT Interface (CATV5)** is enhanced to enable specifying three component names per setup.

The **CATIA-to-VERICUT Interface (CATV5)** is enhanced to output a message when no operation has been selected for transfer.

The **GibbsCAM-to-VERICUT Interface (GibbsV)** is enhanced to enable specifying a Setup Template when the “Append to existing setups in project template” is toggled on. GibbsV will append the operations in the GibbsCAM part file, to the setups that are already defined in the specified setup template file.

The **Mastercam-to-VERICUT Interface (MCAMV)** is enhanced to create profile cutting tools in VERICUT from Mastercam tool descriptions based on the following rules:

- If the shank diameter is same as cutter diameter MCAMV outputs a parametric cutter with the shank correctly represented.
- If the shank diameter is different from the cutter diameter, and the cutter is not a drill, MCAMV outputs an SOR, again with the shank correctly represented.
- If the shank diameter is different from the cutter diameter, and the cutter is a drill, MCAMV outputs a parametric drill, with a holder named "Shank".

The **Mastercam-to-VERICUT Interface (MCAMV)** is enhanced so that the whole Mastercam Interface window fits on the screen when it opens.

The **NX-to-VERICUT Interface (NXV)** is enhanced so that the Gage Point is populated by the following rules:

- Option 1: Use the X offset and Y offset values if specified to populate the Z and X Gage Point values.
 - Offset X in NX is the VERICUT Tool Manager Gage Point Z value
 - Offset Y in NX is the VERICUT Tool Manager Gage Point X value
- Option 2: Use the X Mount and Y Mount if specified.
- Option 3: Use the L, Length, value of the Holder definition if Option 1 and 2 don't exist

The **Pro/E-to-VERICUT Interface (PROEV)** is enhanced with a new feature called **Insert Mode Stock Representation**. This new choice list resides in the “Process” dashboard tab. If “**Before**” is selected, then the “Insert Here” will be moved right before the first Material Removal Feature. If “**After**” is selected, then the “Insert Here” will be moved right after the operation currently being processed. If “**Manual**”

is selected, the “Insert Here” is not moved. This allows you to manually move the insert.

G-Code Processing

New macro, **CycleTurnEndFaceBoxValueK**, is added to support the G94 K value.

Support is added for Siemens 840D commands FLIN and FNORM.

The **BlockSkipAnywhere** macro is enhanced so that the slash (/) character can be used for both block skip and divide. If Override Value = 0 and the slash (/) character is placed at line start, or just after the sequence number it is interpreted as a Block Skip. If the slash (/) character is placed anywhere else in the block it will be interpreted as a divide.

The RotaryDir... macros (RotaryDirCCW, RotaryDirCW, RotaryDirLinear, RotaryDirPosCCW, RotaryDirPosCCWAbsolute, RotaryDirPosCW, RotaryDirPosCWAbsolute, RotaryDirShortestDist, RotaryDirShortestDist180CCW, RotaryDirShortestDist180CW, RotaryDirShortestDist2) are enhanced to support an Override Text value of TURRET. Setting the Turret component to be "Turret" (not "A Turret"), you can now call the RotaryDir... macros passing in a Text Override of "TURRET" to control the rotation of the turret.

Word Type =Special Subtype = Sin840D REPEAT is enhanced to accept "P" or "P=".

New macros **MountCompName**, **MountToCompName** and **UnMountComp** are added to simplify the configuration of head changers.

Driving a Tricept machine given the three actuator lengths is now supported.

1. The following macros are added to support this feature:

TriceptDriveActuators
TriceptActuator1
TriceptActuator2
TriceptActuator3
TriceptAngleA
TriceptAngleC

2. The **Ijk2AbcType** = 34 must be set in Events > Reset >Subsystem

A call to macro **ProcessCutterComp** now updates the setting of the **Process Cutter Comp** feature in the Configure Setup menu: G-Code tab.

A new macro, **WorkingPlane2AbcVertRiderRule**, is added to support the case where the tool is vertical and the Z-Axis and X-Axis are both given.

When cutting in CSS mode, VERICUT now correctly outputs an error when the feed rate is above the specified IPM Fast Feed value.

Error checking is now enhanced to look at the PolarLinearAxis, not just Y when activating Polar Interpolation mode.

If in the XY plane and PolarLinearAxis is X, then Y will be checked for zero.
If in the XY plane and PolarLinearAxis is Y, then X will be checked for zero.

The part for a specific project file now cuts correctly regardless of the of the Animation Speed setting.

Machine Simulation

A **Reset** now restores the original component color if it was changed using the **SetComponentColor** macro.

In the **Machine Settings window: Collision Detect tab**, the checkbox setting no longer changes unless specifically picking in the checkbox.

OptiPath

A new feature, **Tool Wear Limits**, enable you to specify limits for tool wear based on time, distance and volume of material removed. OptiPath will track these values for each tool and when the values exceed corresponding max limit settings, warning messages are issued to the logger and log file.

OptiPath is enhanced to optimize rotated tool path positions (e.g. Fanuc G68) by not applying the rotation matrix to the optimized tool locations. The G68 handles the rotation.

A new feature, **Add Date/Time Stamp to Optimized File** is added to the OptiPath Control window: General tab enabling you to specify whether or not to add a Date /Time Stamp to the optimized file.

Reviewer

The **Line Display** setting (On/Off) is now saved in the preferences file.

Tool Manager

The ability to turn the visibility of individual tool components in the Tool Manager Tool Display area on and off is added.

A Zoller Database Access window enables you to retrieve tool data from the Zoller Database Service and import selected tools from a setup sheet in the Zoller database into the VERICUT Tool Manager.

A new feature, **Tool Component Visibility**, is added to Tool Manager to enable turning the visibility of tool components in the Tool Manager Tool Display area on and off.

The cutting edge of any sweep profile used for a cutter is now highlighted, making it easy to see the cutting "edge" used by VERICUT.

Support is added for setting driven points for TriTools.

The ability for Tool Manager (and Model) sketcher to be able to modify profile arc radius, and have arc begin/end points auto-adjust to stay tangent is added.

In Tool Manager, clicking on the drop down in the Description field of any of the OptiPath records displays all the OptiPath records, both orphans and those that are assigned to other tools.

The following new options are added to the Model File and CAD Tool Insert Cutter definition windows:

Provide the following values:

- 1) Qualified Dimensions
- 2) Driven Point position relative to radius center (9 compass position)
- 3) Tool Nose Radius

VERICUT will create Driven Point, Cutter Compensation and Tool Nose Compensation offsets from the settings.

The ability to input the lead tolerance of a tool tap with a decimal point is added.

The ability to use shortcuts **Ctrl C**, **Ctrl V** and **Ctrl Z** is added to Tool Manager.

Verification

The 7.3 model mirror logic is enhanced. The limitations and restrictions for mirror operations of Pre - V 7.3 releases are eliminated. Pre-7.3 **Mirror About** and **Mirror Relative To values** will be read and converted to use the enhanced features.

Mirror About and **Mirror Relative To** are now transient operations and are no longer stored in the project/machine file.

The **Position** and **Rotation** values are no longer written to the project/machine file. Instead the model matrix is written.

```
<ModelMatrix>  
<MatrixXAxis X="float#" Y="float#" Z="float#"/>  
<MatrixYAxis X="float#" Y="float#" Z="float#"/>  
<MatrixZAxis X="float#" Y="float#" Z="float#"/>  
<MatrixOrigin X="float#" Y="float#" Z="float#"/>  
</ModelMatrix>
```

Pre-7.3 **Position** and **Rotation** values will still be read and supported.

VERICUT is enhanced to enable using multiple **Stop At** points (Breakpoints).

The Project Tree Side Tool Bar can now be hidden and re-displayed, for better Project Tree navigation.

The Project Tree feature that enables you to double-click in a branch in the tree to display a window (for example, to display the Tool Manager, a file selection window, etc.) is enhanced so that VERICUT better differentiates between a simple click and a double click of the mouse button on the branch.

A new feature, **View Components**, is added to Component Visibility in the right mouse button shortcut menu accessed from a Workpiece or Machine/CutStock view. This feature displays the View Components window that enables you to change multiple component visibility settings at the same time.

In the Project Tree a new feature, **Add New CSYS**, is added to the Coordinate Systems branch right mouse button menu.

AutoSave truncation using "name[s]" format is enhanced to work with both upper case and lower case letters inside the brackets. Now [lIiSsFfLlTt] are valid for IP and Image files and [lIiMmSsFfLlTt] are valid for VCT files.

New feature, **Color Theme** (View menu > Color Theme), enables you to select a "favorite" color for the display of the VERICUT Main window and dialogs. This feature is only available when Look & Feel (View menu > Look & Feel) is set to Windows.

Full support (display, X-Caliper, NC Review, IP file) is added for displaying turning fixture profiles in the Profile view.

The ability to automatically output an In Process file, a View Capture image, or a VERICUT Solid file at a specific text string is added to the AutoSave window and the VERICUT Output Files window.

After selecting the Step button, the space bar on the keyboard can be used to continuously step through the NC program for as long as the space bar is depressed. Releasing the space bar will stop the stepping.

In X-Caliper, you can now pick on a cylindrical tool to find a Circle Center in a Workpiece view.

In X-Caliper, you can now select a plane of the cut stock model in a Machine/Cut Stock view.

VERICUT now keeps the Tool ID of Tool Change By File Name when replacing an NC program file in the Project Tree.

The ability to display the pitch of a tap tool in a report is added.

The ability to hide and re-display the Side Tool Bar in the Project Tree is added.

The position of the Center of Rotation Marker is now updated as soon as the Position values are changed.

The Ijk2AbcType macro is enhanced so that when Ijk2AbcType is 34, the following Siemens variables are updated as shown below:

```
$AA_IM[0] = value of the Z register for subsystem 21  
$AA_IM[1] = value of the Z register for subsystem 22  
$AA_IM[2] = value of the Z register for subsystem 23
```

A new Feature, **Add New CSYS**, is added to the Coordinate System branch right mouse button menu in the Project Tree.

An optional register "suffix" is added to VCPPost.

All tool reference models shown in the File Summary window are now in a separate group to enable checking/unchecking all Reference model files at the same time.

Miscellaneous

A new license feature is added to enable a user to return the following licenses to the license server when he is finished with the license without leaving VERICUT:

- Multi-Axis
- OptiPath
- AUTO-DIFF
- Model Export
- Machine Simulation
- Probing
- Cutter Grinder Simulation
- 840D VNCK
- NX Interface
- CAD Model Interfaces

Once an optional license is returned it may be checked-out immediately by another user, including the same user who just returned it.

A Warning or Error message is output to the logger when running a 4 or 5 axis simulation without a Multi-Axis license. A warning is issued when in the rapid mode. An error is issued when in non-rapid mode.

A new feature, **Fillet** is added to is added to the Profile Sketcher to enable modifying a profile arc radius, and have arc begin/end points auto-adjust to stay tangent.

A new feature, **Chamfer** is added to the Profile Sketcher enable creating chamfers without having to “trig” out the appropriate values.

To assist in debugging sync logic, the following 2 new debugging statements are added.

- Debug: Time associated with new Motion/Dwell = 0.09
- Debug: Time remaining for motion on Channel 0 = 0.09

These debugging statement are turned on using the Debug Timing option on the G-Code Output Options: Debug tab (**Project menu > G-Code Output Options: Debug tab**).

The library Heidenhain 530 control is enhanced as follows:

- Support of CIRCULAR PATTERN (Cycle 220) and LINEAR PATTERN (Cycle 221) is added.
- The ANGLE word, used with CYCL DEF 13.1 to Orient the Spindle, now uses the **SpindleOrient** macro.

- The WINKEL word, use in drilling cycles is now supported.

The **MacroVar** macro is added to the library Fanuc control files for M10 and M11.

Machine Simulation Speed is enhanced for particular project file.

A new option, **Fit to View**, is added to the right mouse button menu in the Setup Plan view table. This option will re-size the setup plan image to fit the current view, keeping the plan size ratio and the orientation.

The model mirror logic has been enhanced to allow for limitless mirror operations with no restrictions. If there are an odd number of mirror operations performed on a model, then the model matrix will be left handed. If even, then right handed. There is a new check box, called “Mirrored”, in the model Translate, Rotate, Assemble, Matrix, Csys and Mirror tabs. When checked, it indicates that the model matrix is left handed (odd number of mirror operations were performed). When unchecked, it indicates that the model matrix is right handed (even number of mirror operations were performed).

The ability to check for Near miss between holder and Cut Stock, and Near miss between Shank and Cut Stock is added.

A new option, **Clear Log File on Reset** is added to the Preferences window.

A new feature, **Wedges** is added to the Section window to enable sectioning by angle.

The ability to use a combination of Stop At features and/or text strings is added.

Right mouse pan and middle wheel zoom is added to Sketcher.

Support is added for different Q values in a canned cycle (Fanuc).

A toolbar icon is added for the Output Options window.

The ability to Add/Remove an icon separator is added to the View Toolbar window for use during Toolbar customization.

The Up and Down arrow keys can now be used to step forward and backward in NC Program Review, NC Program Preview and the Reviewer.

Problems Resolved in V7.3

CAM Interfaces

The Mastercam-to-VERICUT Interface (MCAMV) now names selected models based on the current "File Name".

The Mastercam-to-VERICUT Interface (MCAMV) no longer outputs the wrong holder shape and position for lathe tools.

The Mastercam-to-VERICUT Interface (MCAMV) now transfers left hand turning tools correctly.

The NX-to-VERICUT Interface (NXV) now creates drilling tools in the correct orientation.

The NX-to-VERICUT Interface (NXV) no longer strips out subroutines from the project template file.

The NX-to-VERICUT Interface (NXV) now outputs SOR profile models in the correct XZ plane when using the Curve Selection option.

The Pro/E-to-VERICUT Interface (PROEV) now enables defining Cutter Compensation and Driven Points.

AUTO-DIFF

An AUTO-DIFF comparison with the design fully within the cut stock now works correctly in a Profile view.

The Constant Gouge Check On/Off messages are now output to the Logger to show the status of Constant Gouge/Excess Check. The following messages will be displayed as appropriate:

 "Constant Excess Check turned on"

 "Constant Gouge check turned on"

 "Constant Gouge Check /Excess check turned off"

AUTO-DIFF Profile on turned parts now correctly finds gouges or excess.

G-Code Processing

The **CycleTurnRoughCut** macro now works correctly for G64 on NUM Control.

Okuma work offset arrays now supported.

The OSP 7000L G33 CycleTurnThread now removes material correctly.

False probe tip collision errors are no longer output while the probe active when processing a specific Sync project file.

Mirroring now works correctly with Heidenhain polar motion.

A T0M6 command no longer produces an extra, unwanted entry in the Tool Summary in both the VERICUT Logfile and the Tool Report.

VERICUT now re-scans all files at Reset when an init file has been modified.

The **After Motion** setting now works correctly for the **LockComponentAxis** macro.

Project tree Work Offsets created via NC program commands are now removed by a Reset.

False Cutter Compensation inverse motion errors are no longer output after a Heidenhain TNC530 APPR CT (Approach Cutter Tangent) command.

The **Add NC Subroutine Files** button in the Project Tree, Configure NC Subroutines menu now works correctly for multi-setup project files.

Using the **AdjustToolOffset** macro with an Override Value of 2 for a G43 now works correctly with a Right Angle Head.

An M69 LockAxisOnOff now processes correctly on a rotary move line.

The VERICUT session no longer hangs when processing a specific project file with a configuration problem.

Using variables with the **DefineCsys** macro now works correctly.

The **CutterCompFull** macro now works correctly when processing a specific project file.

The cutter comp FULL logic no longer produces invalid circle errors while processing a specific project file.

The **OptiPathOptimizeSubs** macro now correctly outputs the OPTIPATH EXPANSION: comment to the optimized file.

Using the **CutterCompFull** macro in a specific project file no longer causes the VERICUT session to "hang".

The NC Program Window display is now correct when it is opened on a block with a call to an external subroutine, or returning from an external subroutine.

G&L NumeriPath 8000 Local variables not specified now default to zero.

Using the Search feature in the Variables window now correctly finds "local" variables.

The Ijk2AbcType macro is enhanced so that when Ijk2AbcType is 34, Siemens local variables, \$AA_IW[0], \$AA_IW[1] and \$AA_IW[2] are correctly updated.

Machine Simulation

False "CUTTER gouged the Sub_Stock" errors are no longer output when processing a specific project file.

A false "Tool spindle spinning in wrong direction for tool "10" loaded in component "Tool" error is no longer output when processing a specific tapping operation.

Incorrect Overtravel limit errors are now correct.

OpenGL Translucency now works correctly without having to save and then re-open the machine file.

Collision error is now correctly output for a specific project file when spinning chuck jaws collide with the steady rest.

The display is now correctly updated when the **SetComponentColor** macro is used in non-OpenGL mode.

In Process files created while processing a specific multi-setup project file now open correctly in VERICUT.

The **Touch** macro no longer causes a false collision error when using a tool or its holder against the Cut Stock.

The **Touch** macro no longer causes a false collision error with a Mill tool and a Fixture. Axis/Component unlock no longer fails when the unlock command and motion with that Axis/Component are on the same block.

G72 CycleTurnRoughFace no longer leaves 0.001 inch extra without a finish pass.

VERICUT now correctly detects a collision while processing a specific project file.

OptiPath

Optimization results are now consistent whether optimizing multiple tool paths as individual tool paths or optimizing multiple tool paths in a toolpath list.

VERICUT-OPTIPATH comment record now works correctly.

The message "No OptiPath records found for any tool" is enhanced to "Optimized time not available". In the Project Tree, when the Configure Setup menu > G-Code tab > **Apply Acceleration to Cycle Time** is toggled on, optimized time calculation is not available.

The calculation used for the OptiPath time reported in the Status window is enhanced to provide a more accurate estimate when the feature "Apply Acceleration to Cycle Time" in the Project Tree Configure Setup menu: G-Code tab is active. When "Apply Acceleration to Cycle Time" the following warning is output, "Warning: With "Apply Acceleration to Cycle Time" enabled you must run optimized files to get the correct optimized time. The OptiPath Time shown in Status is an approximate estimate".

OptiPath Air Cut feedrate is now correct with Cutter Comp turning off with G40.
IX, IY, IZ, IA, IB, IC programming is now supported for OptiPath.

Tool Manager

When modifying standard cutter shapes, VERICUT now retains parameters that are common to both cutter types.

When using Z Shift in Tool Manager, when you exit without saving, you are now prompted to save the file when you exit the tool manager.

When you attempt to export a tool library loaded in a brand new project (one without a control or machine specified) the "File Summary" tool now displays the appropriate files.

Verification

A problem in a specific project file that causes tool motion to stop when Cutter Compensation is active is now fixed.

The Stock position and rotation values in a specific project file are now maintained when the project file is saved.

Near Miss now works correctly with Insert Cutters.

Near Miss now works correctly for a specific STL tool holder.

The file name of the NC program is now correctly updated in the Status window.

IC is now works correctly with the axes X1, Y1, Z1 in a specific mill turn project file.

When a hole is created with helical interpolation, using the Circle method can now locate the center of the hole.

The Heidenhain CYCL "5" now works correctly for planes other than the ZX plane.

The Project Tree Side Toolbar icons no longer overlap the branch plus (+) signs.

Machine is no longer updated when 'No Machine Animation' is toggled on and Driven Point Zero axes are visible.

The cut color displayed in the Tool Use graph now works correctly when using multiple NC programs.

In the NC Program window when simulating a multi-channel machine, the 'Current' icon is now updated correctly when opening another Project file.

All 4 views are now correctly displayed after saving, and then re-opening, a specific project file.

Components are now correctly positioned after saving, and then re-opening, a specific project file.

A problem causing NC Program Review and X-Caliper not to work after reading 315 sub-programs is fixed.

The Report Preview pane now updates immediately when you change the alignment of an image used in a Report Template.

A false "B axis is locked" error is no longer output for a specific project file.

Right Angle Heads are now supported with 5 Axis rotary head machines.

When changing a Machine components visibility from **Machine View** to **Both Views**, the icon indicating the visibility status of the component now displays correctly.

VERICUT can now access NC program files in a folder with a desktop shortcut.

VERICUT can now access model files in a folder with a desktop shortcut.

Dragging models from Windows Explorer to the Project Tree now invokes red "Busy" light while the model data is being read and the display is updated.

Material removal for a specific project file is now correct.

The Status Indicator Lights now work correctly in NC Preview mode.

A missed collision, while using an offset cutter, is now correctly detected in a specific project file.

SetComponentVisibility now updates the display immediately for VERICUT and NC Program Review.

Material removal for a specific project file using an offset cutter is now correct.

The check for "Missing word before an address" is no longer missing in the Syntax Check window.

VERICUT now outputs valid fast feed error messages when the simulation is using the Configuration menu > Control Settings > Motion tab > Default Feedrate value.

For a specific project file, the correct "Stock" is displayed when changing the "Display Stocks" from one stock to another when the Workpiece and the Machine/Cut Stock views are displayed at the same time.

Facing head material removal is now correct when using a facing head tool with STL inserts.

The activation and de-activation of the X-Caliper window is now consistent when using the Mouse Pick Indicator icon.

Unexpected VERICUT termination no longer occurs when zooming in on the tool using the middle mouse wheel when in NC Program Review mode.

The Cut Stock display is now correct for a specific turning project file.

A false "Error: Stock spindle "Main_Spindle" spinning in wrong direction ..." message is no longer generated when processing a specific project file.

A specific referenced probe tool now contacts the stock correctly in a specific project file.

Support is added for Orivect mode for processing a 5-axis notion using the new RPCP with the input in local/machine coordinates rather than in part coordinates.

Spindle direction is now correct when using a specific Alternate Cutter Tool.

The Surface Speed value shown in the Status Window is now correct when processing a specific project file.

Miscellaneous

A specific Import CAD Tool Data - Holder I snow oriented correctly after revolving.

The directories and files for the CATV icon are no longer missing from the installation.

Support is added for Parameter Q355 (number of thread revolutions) in the library hei430.sub.

The P_PROG_RETURN subroutine, used with sin840d controls now processes correctly when used in a specific project file.

When using a Tool List to populate a Tool Chain, all tools in the Tool List are now visible in the Tool Chain.

STL model export now creates a much better quality model.

Stock transition no longer checks out a Multi-Axis license.

A problem causing the Project Tree slide bar to occasionally not be available is resolved.

OptiPath Learn Mode now works correctly for a specific project file.

An invalid syntax error is no longer output for a specific block.

The window title text is added to the language resource file for all converters (Bincl2v, PolyFix, Sur2stk, Iges2v, and VDA Converter).

The Tool Tip coordinates in the Status window are now reported correctly for milling tools with inserts.

Cutter Grinder now provides access to Cutter Grinder Help.

Setup Plan Symbol images now maintain the correct aspect ratio.

A CSYS can now be repositioned regardless of the method being used to reposition it or the method used to create it.

In the 'Setup Plan', the correct point is now used when using a CSYS Origin to define a Dimension.

Analysis > Inspection > Report template > Table Inspection Features now correctly displays symbols in the preview panel, and in the report when generated.

A specific sweep profile now displays correctly when used in VERICUT.

New Macros in V7.3

CycleTurnEndFaceBoxValueK

FeedChangeMode

HeidSysID

HeidSysIndex

HeidSysNumber

HeidSysRead

HeidSysValue

HeidSysVAR

HeidSysWrite

LinkOffsetFormula

MountCompName

MountToCompName

RotaryDirShortestDist2

SetRobotSingularityTol

SetRobotTurnFlags

Siemens_P_PFRAME_FI

Siemens_P_PFRAME_RT

Siemens_P_PFRAME_MI

Siemens_P_PFRAME_SC

TriceptActuator1

TriceptActuator2

TriceptActuator3

TriceptAngleA

TriceptAngleC

TriceptDriveActuators

UnMountComp

WorkingPlane2AbcVertRiderRule

VERICUT 7.3.1 Interim Release

Release Notes

March 25, 2014

VERICUT Version 7.3.1 is available for all supported Windows platforms. V 7.3.1 contains everything described above for V7.3, plus the following additional fixes/enhancements.

CAM Interfaces

CATIA-to-VERICUT Interface (CATV)

A new PPF button in the lower left corner of the CATIA-to-VERICUT Interface window enables you to save all current CATV settings in the active PPR document. You still need to save that document from CATIA.

CATIA-to-VERICUT Drilling and Fastening Interface (CAT2VDAF)

Template script "AllVisiblePoints.CATScript" is enhanced to display a running total of fastener locations in the CATIA's status line.

Parameters added to the CATIA CATProduct by the CATIA-to-VERICUT Drilling and Fastening Interface are now added to a parameter set called "CGTECH".

Edgecam-to-VERICUT Interface

The Edgecam-to-VERICUT Interface now passes the Design model correctly aligned with the Stock.

GibbsCAM-to-VERICUT Interface (GibbsV)

The GibbsCAM-to-VERICUT Interface now keeps the sub-programs in the Template file when it creates a new VERICUT project file.

Mastercam-to-VERICUT Interface (MCAMV)

Mastercam Center Drill and Holder definitions in a specific project file are now passed correctly through the Mastercam-to-VERICUT Interface to VERICUT.

Changes to the Working Directory in the Mastercam-to-VERICUT Interface, VERICUT Interface window are now applied immediately.

The Mastercam Gage Point and left handed lathe tools in a specific project file are now passed correctly through the Mastercam-to-VERICUT Interface to VERICUT.

The Mastercam-to-VERICUT Interface now automatically transfers any coordinate system that is referenced in the MCAMV, VERICUT Interface window. For example Work Offset Csys, Model CSYS, and Stock Csys.

The ability to multi-select Programs and subprograms in the Mastercam-to-VERICUT Interface is added.

The Mastercam-to-VERICUT Interface now correctly transfers the Work Offset and driven point from Mastercam to VERICUT.

A MERGE keyword for the mini-project file (*.vctmp) is added in the OPS file. It means that G-Code tables, NC Programs, G-Code Subroutines will be merged instead of removing G-Code offsets already configured in the Template file.

NX-to-VERICUT Interface (NXV)

The ability to extract a solid model assembly from NX and pass it through the NX-to-VERICUT Interface into VERICUT is added.

New option, "Create One Model File for Multiple Stock Selection" in the NX-to-VERICUT Interface, VERICUT Interface Options window enables you to pass Stock models as one model or as many.

The determination of the gage point value for turning tools passed through the NX-to-VERICUT Interface is enhanced as follows. If the X and Y offsets are set to zero, then don't do anything. If a value is specified for the NX Tool X offset, have that value transferred to the VERICUT Tool Manager Z Gage Point value. If there is a value for the NX tool Y offset, then transfer that value to the VERICUT Tool Manager Gage Point X offset.

The NX-to-VERICUT Interface now supports NX 9.0.

The NX-to-VERICUT Interface now outputs the shank on user defined tools.

The NX-to-VERICUT Interface now considers shank from the NX shank tab for 10 parameter milling cutters when calculating the gage length.

The NX-to-VERICUT Interface now uses a default lead tolerance value of 1% for Tap tools.

Pro/E-to-VERICUT Interface (PROEV)

A problem preventing the Pro/E-to-VERICUT Interface from opening when VERICUT is running with a single platform license is fixed.

The Pro/E-to-VERICUT Interface no longer creates a tool library in the working directory with the same base name as the project file, with all its associated tool/holder models when the option "Use Tools from the Setup Template" on the Options Panel is selected.

The Pro/E-to-VERICUT Interface now supports machine file names with special XML sub-strings. The Pro/E-to-VERICUT Interface can only find machine files if they reside in the same directory as the Project Template file or in one of the library paths specified by CGTECH_LIBRARY environment variable.

G-Code Processing

Support is added for a new "painting" mode by a milling tool via a new macro **PaintModeOnOff**. Turning it on will turn the currently loaded milling tool and EVERY milling tool loaded thereafter into a "painting" tool. In this mode, no material is removed from the cut stock but the areas of the cutter/stock contact are shaded with the cut color. The mode must be turned off to restore normal cutting. There is no Tool Manager toggle. The ON/OFF macro can be used mid-tool. X-Caliper > Feature reports both the feature "underneath" the paint and the history of the "painting" record.

New macro, **CutterCompValueWarning** is added to output an Error or Warning on unreachable motion segments. Use with Override Value=1 to produce a Warning message and Override Value=2 to produce an Error message, on unreachable motion segments. This macro should be called after macro **CutterCompFull** with an override Value=1, and will be effective only in the full mode of cutter compensation. The messages are: "Compensation value too large: cannot offset circle with radius = r" and "Compensation value too large: motion block cannot be reached.

Programming a rotation with Cutter Compensation active now works correctly in all three modes of cutter compensation: the full mode (CutterCompFull, Override Value=1), the 3D mode (Cut3dcMode, Override Value=1), and the standard look-ahead mode. Please note that the full mode setting is ignored when in 3D mode.

Clockwise arcs in a specific project file are now offset correctly when cutter compensation is active in 3D mode.

A problem resulting in "Error: Cutter Compensation caused inverse motion" messages when cutter compensation Full mode is active, is fixed.

New macro, **ToolAxisMotionXYZ**, is added to support the Siemens "MOVT=IC" command. This macro will cause the tool to move the specified distance along the tool axis. This macro only works with a simple XYZ movement (no UVW, no reverse axis, axis must be orthogonal, ...).

Using macro, **SiemensSystemFramesCancel** and then macro **SiemensSystemFramesRestore** no longer results in undesirable motions when used with a SUPA command.

The part in a specific project file is no longer gouged, after using a SCALE function while in Circular Interpolation mode, while processing very large arcs.

Cutter Compensation with a changing offset parameter in a specific project file now produces the desired result using a Siemens 840D OFFN command at cutter compensation to indicate a variable compensation value.

The tool axis vector is now calculated correctly for a water jet cutter.

The Frame link is now calculated correctly when using the SiemensCondColon conditional.

New option **Debug Control Subs**, is added to the Output Options window, Debug tab. With this switch turned on (Yes), no messages are suppressed from the G-Code log file while processing control subroutines. Although this switch goes against the general concept of control subroutines being black boxes. It is a way to get quick access to this information without moving all the subs. The default (No) is to suppress this information when processing control subroutines.

A new error message is now output when a **CoupleAxisOn** or **CoupleAxisOff** macro fails.

A second G41 Cutter Comp call no longer fails to display yellow arrow in NC Program window.

Both B axis rotations now stop at the correct position in a specific project file.

The **LinkOffset** macro is enhanced with an "IMMEDIATE" text option. When Override Text=IMMEDIATE, when the **SetAxisCompLink** macro is called, the offsets will be updated immediately.

The **Siemens840DSyncBegin** macro is enhanced to work with `_MPF` as well as `_SPF`.

The **SiemensMEAFRAME** function is enhanced to behave like the MEAFRAME command on the machine.

The `$P_ACTFRAME` is now updated correctly after reading a TRANS or ATRANS command.

An false "Error: Invalid number of arguments passed to HeidCmdIfGreater(), expecting 2" error is no longer output when VERICUT reads a D11 P01 Q5 P02 Q6 P03 1 line.

A false "Cycle expand" error is no longer output for a specific project file.

Support is added for the Sin840D command, `$P_CYCFRAME`. New macro, **Siemens_P_CYCFRAME** is added to be used on variable changes. Macros **SiemensSystemFramesCancel** and **SiemensSystemFramesRestore** are enhanced to support text CYCFRAME.

Unexpected VERICUT termination no longer occurs when clamping the CutStock back to the Fixture after the bar feeder acts.

The Z-axis no longer moves when Polar Interpolation is cancelled.

False "Array size" and "Array index" errors are no longer output for a specific project file.

Unexpected VERICUT termination no longer occurs when processing a Sin840D cycle800 command in a specific project file.

The logical equal '==' in a specific project file now processes correctly regardless of the order of the 2 elements.

The Okuma OSP OkumaGotoJump G14 Sync no longer jumps up into G13.

The syntax check results for a specific project file are now consistent with the results obtained in V7.2.

The Hei530:Tool DEF in a specific project file is now correctly selecting the tools.

New macros **ToolChainTimes**, **ToolChainPreSelect**, **ToolChainPreSelectID**, and **ToolChainToolChange** are added to enable getting more accurate cycle times for machines that have a tool chain. A new Tool Chain debug option is added to the Output Options window: Debug tab to assist in debugging Tool Chain problems.

The **CyclesTap** macro now reverses the spindle at the bottom of the hole and the tap feed direction is monitored in addition to the spindle direction. VERICUT is now sensitive to feed (along tool axis) and spindle direction for tapping tools.

Machine Simulation

A specific encrypted machine file now loads correctly.

The VERICUT simulation no longer hangs/freezes if you attempt to rotate the machine view (e.g. via left-click and drag mouse) or zoom while the machine is executing the tool change subroutine.

Cutter compensation 3D mode (Cut3dcMode, ORV=1) is enhanced to enable changing the orientation of the Rotation Plane Zero while Cutter Compensation is active.

The simulation of a specific project file no longer reports an error that the Probe Tool has exceeded the near miss value when Machine Collision Detection is not active.

Having the MDI window open while cutting no longer causes the display of the cut stock to intermittently disappear from an accelerated OpenGL Machine/Cut Stock view.

Unexpected VERICUT termination no longer occurs for a specific project file when typing the word ;(ACESSORIO=T30606) in the MDI window.

The VERICUT session no longer hangs for a specific project file when Collision Check between Active Tool and Stock is active.

False "Warning: macro is undefined" messages are no longer output for a specific project file when processing CGTECH_MACRO when using a MAKKA pre-processor.

A specific project file now cuts the part correctly when using bull nose cutters.

In a specific engraving project, the TRANS command now moves in the correct direction when following a CYCLE800 command.

OptiPath

OptiPath no longer modifies the IJ values of the circle center point, when optimizing a circular motion when there is an error in the initial point radius and the terminal point radius.

OptiPath no longer removes G65 blocks, while optimizing a specific NC program, when the simulation is paused inside a control subroutine.

OptiPath no longer strips out subroutine calls, from a specific NC program, when optimizing with Stop at Collision is toggled on.

The VERICUT log file now shows the correct time savings for optimization when using VERICUT-OPTIPATH comments to change the optimization settings.

The VERICUT log and VERICUT report files now show multiple OptiPath records when using VERICUT-OPTIPATH comments to change optimization settings during cutting operations. All OptiPath settings used during a tool sequence are now listed in the order in which they are used.

OptiPath now recalls the initial feedrate after a VERICUT-OPTIPATH OFF command. OptiPath now adds an Fxxx line with the programmed feedrate BEFORE the (VERICUT-OPTIPATH OFF) line in the optimized file.

False "Invalid circle" errors are no longer output when simulating the optimized NC program for a specific project.

An NC program optimized using the Circle Feedrate = Break-up setting no longer outputs false "Invalid circle" errors when simulating the optimized NC program for a specific project.

OptiPath no longer adds Air Cut Feedrates when VERICUT-OPTIPATH OFF and VERICUT-OPTIPATH ON comment records are used with Cutter Compensation mode set to "full look ahead".

An NC program optimized with Radius Control toggled on no longer modifies the toolpath when simulating the optimized NC program for a specific project.

OptiPath no longer reports false "Cutters with concave profile are not allowed" warnings when Constant Chip is used when the shank of a profile tool has a chamfer.

Optimizing subroutines in a specific NC Program file no longer moves tool change blocks out of the main program file and outputs them to subroutine.

VERICUT now shows the correct OptiPath Savings values for a specific project file when optimizing only Air Cut Feedrates.

OptiPath now puts a space between the line number and C Word Address in optimized file.

OptiPath Learn Mode now captures the "worst case" cut condition when the Update Optimization Settings for Existing Records option is toggled on in the OptiPath Control window: Learn Mode Options tab and the same tool is used multiple times in the same NC program.

The correct OptiPath Time is now reported after changing the Volume Removal value then re-optimizing a specific NC program.

Reviewer

VERICUT Reviewer is now available on an iPad.

VERICUT Reviewer now shows a Refined display in the Workpiece view during the replay of the simulation when Replace Material is applied.

The stock display for a specific turning project file is now consistent between VERICUT and the Reviewer.

The display of a specific holder, in a specific project file is now consistent between VERICUT, NC Program Review and Reviewer.

A specific 2 setup reviewer file now displays the tool correctly in the Machine/Cut Stock view during the replay of the simulation in Reviewer.

Unexpected Reviewer termination no longer occurs for a specific reviewer file that was created with 2 Setups, the first Setup being inactive.

The stock and tool display for a specific project file using a machine with a sub-spindle is now consistent between VERICUT and the Reviewer.

Tools in a turret are now displayed correctly in Reviewer at the start of simulation.

The Reviewer no longer opens without the Toolbar displayed. When you choose View > Toolbar, the message area no longer reports a NullPointerException error.

Tool Manager

An alternate Holder in a specific project file no longer displays in a Workpiece View when Spindle spins.

A specific imported STL cutter, in a specific project file, now correctly spins about the spindle axis thereby producing the correct material removal.

The Look & Feel: Nimbus option is enhanced to provide a larger text field height in all windows that have a table with a text input field (Machine Settings, Word Format, Tool Manager, etc.).

Tool Manager is enhanced to enable creating disk, cylinder, hemisphere and extension (flat cylinder) probe tips.

When tool assembly components have been changed in Tool Manager, you will now be prompted with the following message, "Values have been modified. Do you wish to save before the close operation", when closing Tool Manager.

When changing a referenced tool holder in TOOL Manager both the Holder ID and the Description are now both updated.

Special characters used in tool library file names no longer result in "Error loading tool library" errors when the tool library file is loaded in Tool Manager.

The "Automatic Gage Offset Z" setting now works independent of the "Stack" setting. If the "Stack" setting modifies the overall length of the tool and "Automatic Gage Offset Z" is toggled "on" the Gage Point field is also updated.

The Stand-alone Tool Manager now correctly recovers the 'ID' column width from the Preferences file.

Using Ctrl+X, Ctrl+C and Ctrl+V now work correctly when used inside a text field in Tool Manager.

Cutters are no longer missing in a tool library file after importing a specific set of tools from the Lemoine Tool Database.

Unexpected VERICUT termination no longer occurs after using X-Caliper Feature History to on a specific holder in Tool Manager and then closing Tool Manager and doing a Reset in VERICUT.

A specific tool library file is now correct when reopening the tool library file after adding tools duplicated from another tool library, renaming the tools and saving.

A specific tool profile created using the Chamfer feature in Profile Sketcher, now displays correctly when used in a VERICUT simulation.

VERICUT now recognizes the combined flute length when multiple cutters are stacked in a tool assembly.

Now when importing an NX tool in Tool Manager using Import > CAD Tool, a Windows "wait" icon is displayed until the translation is finished, then if successful, the part geometry will load automatically.

The DynamicToolTip control point no longer moves when a specific duplicate tool is used.

Verification

The "Waterjet reentered material at line xxx" error message is changed to a warning message.

False warning message "Arc start and end points may produce undesirable motion on Fanuc controls ..." are no longer output for a specific project file.

Unexpected VERICUT termination no longer occurs while processing a specific project file with a bad "sync" configuration.

During the scan pass, VERICUT now associates branch points based on whether it is associated with G13 or G14. The basic structure has been added to support this concept with any of the sync controls, but currently it is only implemented for Okuma, G13/G14. NOTE: This works for both branching to labels, and branching to sequence numbers.

When looking for a subroutine, VERICUT first looks for a matching sub ID and then if none is found VERICUT will try to find a matching filename.

A simulation recorded using the VERICUT Movie feature can now be played back using the Playback VERICUT Movie feature inside of VERICUT or using the stand-alone VERICUT Player (player.bat).

VERICUT no longer fails to detect a holder collision in a specific project file.

Unexpected VERICUT termination no longer occurs for a specific project file running in a multi-setup environment with auto-saving In Process files, and then merging the In Process Files.

Unexpected VERICUT termination no longer occurs while processing a specific project file that is using the CutterCompFull macro with full look-ahead mode.

The X-Caliper Highlight Same Plane feature is changed to Highlight Same Plane/Cylinder and enhanced to also find same cylinders. Picking on a cylinder feature automatically fills-in the radius field and colors all cylinders with the same radius, within the specified tolerance.

A CSYS now behaves the same as any other object. It can be rotated about the active CSYS, a specified Center of Rotation, or local if checked.

VERICUT is enhanced to exclude the tap stem from holder-stock near miss collision checking.

Creating an AVI recording with OpenGL turned on, now creates a good AVI file.

The **SetDynamicVars** macro, CurSubName option now updates a variable correctly when the variable was set in the init file and created as a number variable and then used as the subroutine name.

The Animation Speed Slider setting no longer affects collision reporting for a specific turning project file.

The Animation Speed Slider setting no longer affects NC Program Preview computing time.

The **SetComponentVisibility** macro now updates the display immediately for VERICUT and NC Program Review.

The Tool Use Graph image that is output into reports now contains the tool ID and time for each tool.

False "Invalid circle statement ..." errors are no longer output for a specific NC program using circular interpolation with a DEP command.

The Volume Removal Rate in the Cutting Conditions Graph now produces the correct results for a specific project file with "Check Cutting Limits" turned on but the tool does not have an OptiPath record.

The tools displayed in the Workpiece view are now consistent with the tools displayed in the Machine/Cut Stock view for a specific project file using a 3 spindle machine.

Sectioning a Stock model before the cut stock is created will now have the section faces colored the specified color.

Three new text variables, Stock Envelope X, Stock Envelope Y, and Stock Envelope Z are added to VERICUT's Report feature to enable separate Stock Envelope X, Y, and Z values.

The near miss tolerance value is now disabled for checks between a probe tip and the fixture.

When a machine file is saved and the units are changed (e.g. from inch to mm) the conditional rotary values no longer have their units changed.

When using X-Caliper, you are now able to select a circle center on the Fixture in a Workpiece view after the cut stock is moved to the second setup position.

VERICUT simulation is no longer extremely slow when simulating a specific CAD profile insert in a specific project file.

Unexpected VERICUT termination no longer occurs when simulating a specific project file.

False "Insert off Turning Plane" errors are no longer reported for a specific project file when Holder/Stock and Holder/Fixture collision checks in the Project Tree are turned off.

Using Syntax Checking for a specific NC program file now produces the same results that it did in V7.2.

VERICUT no longer outputs false limit violation warning messages for tools which do not have OptiPath records when Check Cutting Limits is toggled "on".

The X-Caliper Hole Depth feature no longer fails when measuring a turned chamfer in Workpiece view.

A new Project Notes tab is added to the Properties window enabling you to date stamp / watermark VcProject files in the same way as can be done for machines and controls.

Unexpected VERICUT termination no longer occurs when double clicking on a line in an NC program that precedes a line using variables.

Filter Variables settings, in the Variables window, are now saved when the Variables window is closed so that they can be recalled when the Variables window is re-opened.

The tool display in NC Program Review is now correct for a specific project file using a head changer machine.

Unexpected VERICUT or Tool Manager termination no longer occurs when simulating a specific tap tool in VERICUT or viewing the Tool in Tool manager.

Unexpected VERICUT termination no longer occurs when opening the Variables window or the Control Variable window when a specific project file is loaded that uses a control file with excessively large array variables.

Material removal is now correct for a specific STL based insert tool.

Motion is now correct when simulating a specific project file that uses the CycleMillPocketType macro with the CutterCompFull macro set to full look-ahead mode on.

The tool display in the Workpiece view is now correct when simulating a specific project file.

The machine no longer moves when an axis is selected in the MDI window.

When simulating a specific project file, the insert of a specific tool no longer leaves a trace of the insert in the Profile and Machine/Cut Stock views when the insert backs out from a cut in Rapid.

Creating a Solid of Revolution (SOR) model using Sketcher now works correctly.

The text of the error message that Constant Gouge Check outputs is changed to “Error: Maximum Allowable Gouge violation for tool "*tool number*" loaded in component "Tool" and "Design" at line: ...”.

The Report Filename now defaults to the project file name if not otherwise specified.

The check for spindle direction now works correctly for the situation where a milling cutter is offset from the tool Z-axis and is cutting a turning stock along the center line.

The graphic display is now correct for a specific turning project file which has a situation where both the tool and the stock spindles are spinning, the tool is large compared to the stock and the tool/stock contact area is very small.

The motion in a VERICUT simulation of a specific project file is now the same as the motion on the machine.

An "ArrayIndexOutOfBoundsException" error no longer displays in the Logger after opening a specific In-Process file and then right clicking on the Play button (to edit Stop At Options).

The display of a specific revolved STL cutter is now correct.

A collision between the Holder and the Fixture in a specific project file is now reported correctly.

VERICUT Drill and Fastener Simulation (VDAF)

New feature "Single Layer Fasten Check", in the VERICUT Drill and Fastener Simulation Configure Simulated Locations menu, is added to enable turning on and off single layer checking to enable simulating fasteners that have been intentionally inserted into a single skin layer without errors.

Miscellaneous

Feature Design Display "Translucent" on the AUTO-DIFF window, Settings tab now works correctly.

All Okuma library controls are updated with **XAxisMotionLimit**, **YAxisMotionLimit**, **ZAxisMotionLimit**, **UAxisMotionLimit**, **VAxisMotionLimit** and **WAxisMotionLimit** for rapid only moves.

The Library Siemens 840d control is enhanced to support KNUM = 1 (Work Offset gets updated after probing). The key point is that the MEASURE() function is setting \$AC_MEAS_FRAME based on \$AA_MEAS_SETPOINT values. The MEASURE() function has been enhanced to do so and now _DELTKOR[0] gets calculated correctly and thus \$P_UIFR[] is updated accordingly.

The Library Siemens 840d control is enhanced to restore the spindle status at the end of any proc call. The P_PROG_RETURN subroutine now restores spindle state.

The Library Heidenhain heid530.ctl, IROT ** command (incremental rotation) now remains active until cancelled by a ROT 0 command.)

The Library Siemens 840d control is enhanced so that the \$P_WPFRAME variable is applied immediately as it changes.

A "PLANE SPATIAL ... STAY" command now correctly orients the Driven Point Zero for a specific project file.

The cgtech.lf file is now created correctly after a drag and drop sequence.

The V73 LicenseTool.exe is enhanced to ensure that there is a space after letter *E.

A problem causing a Null Pointer Exception error on a specific computer when opening the File Summary window is now fixed.

The display format of the "Total Time Difference" value in in a VERICUT report is now correct.

The ability to post-process in batch mode is added. To run a post in batch mode use following arguments (in addition to project and "batch") in the command line:

```
POST_SUBSYS=subsystem ID
POST_FILE=filename.vcpost
POST_OUT=filename.out1
POST_LOG=filename.out2 (optional)
POST_RUN
```

To record output(s), argument POST_RUN must be specified. It is equivalent to switch "Run Post while simulating" in the Post Setup window.

In a specific project file using a multiple spindle machine, NC Program Review now displays the CutStock at the correct location when the part is transferred from one Spindle to another one.

Angle head components, in a specific project file, are no longer displayed in the Machine/Cut Stock view as revolving when viewed in NC Program Review.

The Optimized time is now shown in the Tool Use graph regardless of the Apply Acceleration to Cycle Time setting in the Configure Setup menu: G-Code tab. However, you should pay attention to and keep in mind the warning that VERICUT issues when optimization begins that OptiPath times are not reliable with Accel/Decel ON.

Constant Gouge Check now produces consistent results whether G03 motions are used or G01 motions are used, regardless of the Animation Speed Slider setting.

A problem causing a specific set of project files to process much slower in V7.3 than they did in V7.2.3 is fixed.

A tool in a specific project file no longer moves to the wrong position in the first block after ROT while Cut3D is active.

False "Line exceeds maximum number of tokens" errors are no longer output for a specific project file.

The OPS File logic is modified such that if a component name already exists, it is updated with the models from the mini-project file (.VcTmp), but its position in the template component tree is not changed.

A specific engraving project file using an 840D CYCLE60 command with TRANS/ROT when in Polar Interpolation mode now simulates correctly.

The motion in a specific project file using the Siemens 840D TRANS command is now correct.

The simulation of the machine head in a specific project file is now correct.

New Macros in V7.3.1

The following new macros are added for V7.3.1

ApplyStockRotation2ijk

AllowVariableZeroToBeSet

AxisVarsMapping

IfBlockIfCheck

IgnoreTableRotMatrix

PaintModeOnOff

ProcessOpSubstitutions

RpcpMatrixType

SiemensDefArraySize

SiemensDefArrayTotalSize

ToolChainPreSelect

ToolChainPreSelectID

ToolChainTimes

ToolChainToolChange

VERICUT 7.3.2 Interim Release

VERICUT Composites Only

Release Notes

May 5, 2014

VERICUT Version 7.3.2 is available for all supported Windows platforms.

V 7.3.2 contains everything described above for V7.3.1, plus the following additional fixes/enhancements.

VERICUT Composite Simulation (VCS)

Enhancements

VCS is enhanced to read the CSYS's from VCP via the VCP to VCS Interface.

A new "Delete Cut Laminate" feature enables you to delete or keep pieces of cut laminate. This feature is typically used to delete pieces of excess laminate that have been cut free from the workpiece.

VCS is enhanced to enable changing the robot tool tip during simulation. Bi-directional tape laying heads have the ability to move the roller position via NC program command. When the roller position moves the robot's tool point location transforms with it and tape is laid in the opposite direction of the previous roller position, without rotating the head. VCS (and VERICUT) are now able to move the robot tool point in order to correctly simulate this action.

The first robot motion and the rotisserie angle in a "mixed" rotisserie configuration is now correct. The very first motion (if it is RAPID only) is processed as a FROM command. All consecutive motions are processed as programmed (LIN or PTP).

Problems Resolved

Machine simulation now works correctly in VERICUT Composite Simulation (VCS) Limited.

VERICUT Composite Programming (VCP)

Enhancements

The Tow-boundary overlap feature is enhanced to enable you to choose to have no part of each tow go beyond the boundary, to have each tow cut where its centerline crosses the boundary, to have the boundary's interior fully covered, or to have each tow extend past the boundary by a specified distance.

A new feature, "Head display and collision avoidance check" on the Roller and head visualization card, provides a collision avoidance capability which can apply pitch and roll adjustments to the head while post-processing. This is based on an STL model of the head and it attempts to prevent the model from interfering with the form or any prior laid material.

A new option, "Toward outside" is added to the "Minimum tow extension" feature on the Setup card, extends the tows toward whichever end of the course's head path is closer.

VCP is enhanced to enable not laying minimum length tows. For users that do not want to lay-down minimum length tows but instead want to manually apply a triangular patch in corners that cannot be laid with the correct (too small) tows, the following procedure will accomplish that;

1. Create all of a ply's tows, long and short.
2. Tick the box on the Add card labeled "Ignore material in minimum tow extensions".
3. Add material for the ply.
4. Set the tow coloring on the "Color" card to distinguish between long and short tows.
5. Delete all the short tows which will be replaced by patches.

VCP is enhanced to support bi-directional tape laying heads.

Problems Resolved

64 bit VCP no longer requires that a 32 bit version also be installed.

Reading a CATPart that has consecutive space characters in the path name is now supported.

After using the "Reseat Courses and Links" button on the Ply card, and re-posting, the new program now correctly contains the Tow commands and active Tows in the comments.

After importing a specific NX-part (imported step) and a STEP model (original file) to VCP two surfaces are now listed in the surface table and displayed graphically.

The VCP session no longer hangs when generating the first course in a specific ".VCP" file.

VERICUT Composite Paths for Engineering (VCPe)

Enhancements

The "Envelope" card is now exposed in VERICUT Composite Paths for Engineering (VCPe). The table on that card has 2 extra rows for the total lengths of the on-form and off-form motion in the selected work-packet. There are fields under the table where the user can provide figures for the average on-form and off-form head speeds. Cycle time and the lay-down rate are then estimated. Time is in minutes. Lay-down rate is in kg/hr or lb/hr.

CAM Interfaces

CATIA-to-VERICUT Interface (CATV)

CATV is enhanced to exclude information from the CATIA Tool Definition Compensation tab. CATV now checks for the "CGTECH_CATV_DRIVEN_POINTS" environment variable. If it is set to "No", "Non", "Nein", "Niet" or "Nada" you won't get any driven points in the resulting tool libraries.

Mastercam-to-VERICUT Interface (MCAMV)

The Stock in a specific Mastercam 5 file is now passed through MCAMV to VERICUT in the correct position.

NX-to-VERICUT Interface (NXV)

NXV is enhanced to enable specifying the location of the NXV preferences file without changing the %HOMEDRIVE%%HOMEPATH% variable. Environment variable CGTECH_NXV_PREFSDIR is added to specify the location of the NXV preferences file.

Example of use:

Set CGTECH_NXV_PREFSDIR=C:\CGTech72\nx

If CGTECH_NXV_PREFSDIR is not set NXV will attempt to use the preferences file in the default location %HOMEDRIVE%%HOMEPATH%.

G-Code Processing

A new macro, **SetRelationalWorkCoord**, is added to enable creating a relational work coordinate offset from a G10L2 statement.

A new macro, **EndBlock**, is added to enable ending the current block (IF, LOOP or SUB).

The Sin840D DEFINE function in VERICUT now works the same as it does on the machine.

The Sin840D DEFINE function in VERICUT is no longer case sensitive (upper case / lower case).

The Sin840D DEFINE function in VERICUT now looks at the entire word for the variable like it does on the machine.

A specific project file using macro **HelicalAddFullLoops** with macro **CutterCompFull** turned on now produces the correct number of turns and a correct pitch calculation.

The **Type2CLS** macro now produces the correct results when mathematical expressions are used in the Type 2 arguments.

An Okuma number array variable in a specific project file now processes correctly.

Machine Simulation

A specific project file using closed contours now simulates correctly when **CutterCompFull** is turned on.

Unexpected VERICUT termination no longer occurs when processing a specific project file using merged turning with an index chuck.

The rotary index chuck used in a specific project file is now indexed correctly with Sync Waiting.

Tool Manager

The value in a Tool Manager Cutter Compensation record is now left justified to improve readability. The number of decimal places displayed is reduced from 8 to 4.

Changing the insert position for a specific tool in Tool Manager no longer creates an incorrect Qualified Dimensions entry.

Miscellaneous

The 7.3 Composites Installer now has an option for license server only.

New Macros in V7.3.2

The following new macros are added for V7.3.2:

EndBlock

Ijk2AbcSpindleOrient

SetRelationalWorkCoord

VERICUT 7.3.3 Interim Release

Release Notes

October 14, 2014

VERICUT Version 7.3.3 is available for all supported Windows platforms. V 7.3.3 contains everything described above for V7.3.1, plus the following additional fixes/enhancements.

CAM Interfaces

CATIA-to-VERICUT Interface (CATV)

Unexpected VERICUT and Tool Manager termination no longer occurs when certain tools created from CATIA User Representations are passed into the VERICUT tool library.

The CATIA-to-VERICUT Interface now correctly passes a specific tool that has a driven point (corrector in CATIA terminology) with a blank "type" to VERICUT.

The CATV.exe now looks for the CGTECH_SINGLE_PLATFORM environment variable and automatically inserts "Limited" in the command line it uses to invoke VERICUT.

CATIA-to-VERICUT Drilling and Fastening Interface (CAT2VDAF)

The CATIA-to-VERICUT Drilling and Fastening Interface is enhanced to enable mirroring skin, structure or fixture models in the same way as the CATIA-to-VERICUT Interface.

The CATIA-to-VERICUT Interface is enhanced to look for milling tool profile sketches in both part bodies and in geometrical sets when passing 3D-tools from CATIA through the Custom Tool Geometry list in CATV.

The CAT2VDAF.exe now looks for the CGTECH_SINGLE_PLATFORM environment variable and automatically inserts "Limited" in the command line it uses to invoke VERICUT.

Edgecam-to-VERICUT Interface

The Edgecam-to-VERICUT Interface is enhanced to support multiple G-Code offsets Edgecam 2014 R1 and later.

The Edgecam-to-VERICUT Interface is enhanced to support selecting from multiple attach components for Edgecam 2014 R1 and later.

Deleting sequences in the EdgeCAM-to-VERICUT interface now works correctly in the latest EdgeCAM-to-VERICUT interface for Edgecam2014 R2.

Using a horizontal machine and using offsets within the EdgeCAM interface, the coordinates now travel with the stock correctly in the Edgecam-to-VERICUT Interface for Edgecam 2014 R2.

GibbsCAM-to-VERICUT Interface (GibbsV)

The GibbsCAM-to VERICUT Interface is enhanced so that when using the GibbsV to post programs when selecting Generate and Run and the post-processor has an option in it that controls the extension of the output file, GibbsV will read this field from the post-processor and if there is an extension present, the file will be output with that extension. If the field is left blank, then the file is output with no extension.

The GibbsCAM-to-VERICUT Interface now correctly saves the selected Attach Axis System when closing the GibbsV window and then reloads it when the GibbsV window is reopened.

The GibbsCAM-to-VERICUT Interface now transfers all GibbsCAM coordinate systems to VERICUT.

The GibbsCAM-to-VERICUT Interface now correctly transfers 2 specific tap tools, in a specific GibbsCAM VNC file, to VERICUT.

Mastercam-to-VERICUT Interface (MCAMV)

The Mastercam-to-VERICUT Interface now transfers a specific coordinate system from Mastercam into the correct location in VERICUT.

The Mastercam-to-VERICUT Interface now transfers a specific Lathe tool insert with the insert thickness in the correct direction.

The Mastercam-to-VERICUT Interface now transfers a specific Custom tool to VERICUT with the correct units.

The Mastercam-to-VERICUT Interface now transfers a specific lathe tool assembly, consisting of metric tool insert and an inch holder, correctly to VERICUT.

The Mastercam-to-VERICUT Interface now transfers a specific Mastercam tap tool assembly correctly to VERICUT.

The Mastercam-to-VERICUT Interface is enhanced to include a Calculate Relative to Location option in the VERICUT G-Code Tables section of the MCAMV VERICUT Interface window. This feature enables you to have a relational offset recalculated in the machine position where the offset will be used. The new position is immediately calculated and stored and therefore is not dependent on the machine position when the offset is activated.

The VERICUT icon in Mastercam used to start the Mastercam-to-VERICUT Interface now displays correctly when switching to “Large Buttons” using Mastercam’s Settings > Configuration > Screen > Large Buttons checkbox.

The Stock Delta X, Y, and Z text fields, in the MCAMV Options window, have been lengthened from 36 to 60 pixels enabling you to see all of the characters entered in the text boxes.

G-Code Processing

New macro, **AutosetTableAxisArrayVars**, is added to provide support for Okuma OSP BiDirectional Alpha Array Work Offset System Variables.

Offsets are now correctly processed for a specific project file using a Siemens 840D SUPA command.

New macro, **SiemensSetChannelTags**, is added to support a new Siemens 840D syncing method.

Support is added for Fast Feed IPR values. The existing "Fast Feed" label on the Project Tree Configure Setup; Motion tab is changed to "Fast Feed (FPM)". A new field "Fast Feed (FPR)" is added. If in FPR mode and the Fast Feed (FPR) is non-zero, fast feed will be checked against the Fast Feed (FPR) value. Otherwise: The fast feed will be checked against the Fast Feed (FPM) value. For Tapping and Threading, the fast feed will be checked against the Fast Feed (FPM) value.

Macro **HelicalAddFullLoops** now produces correct number of turns with the correct pitch.

New macro, **Siemens840DCallSubName**, is added to enable removing the path and suffixes from a Siemens EXTCALL command.

Adding **CutterCompFull** Override Value = 1 to Events, Start of Processing of a specific project file, no longer causes VERICUT to unexpectedly jump into subroutines that are not called by the NC program.

Using **CutterCompFull** Override Value = 1 with a specific tool in a specific project file no longer results in incorrect motion.

Using Okuma OSP G72 CycleTurnRoughFace now correctly outputs a Fast feed for the G00 on the final pass in a specific project file.

A new option, **Shortest Distance 3** is added to the Absolute Rotary Direction feature on the Configuration menu > Control settings window > Rotary tab. It functions the

same way as the **Shortest Distance** option, but no error messages are output when a 180 degree rotary move is programmed.

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

False "Invalid circle statement ..." messages are no longer output for a specific project file using macros **RpcpOn** and **RpctInput** with Override Value = 1.

When a G96 CSS is active on the part spindle while processing a CYCLE84 tapping cycle on the tool spindle no longer results in false "Error: Feed per rev of "2.1856" does not match pitch of tap tool "13" at line: ..." messages.

Unexpected VERICUT termination no longer occurs while processing a specific project when doing merged turning on a machine with an index chuck.

Processing **IjkAAxisMotion**, **IjkBAxisMotion**, and **IjkCAxisMotion** macros now produces the correct results in a specific project file also using a **CoupleAxisOn** macro.

The macro **SiemensSystemFramesCancel** now cancels all Frames including \$P_TOOLFRAME. The **SiemensSystemFramesRestore** macro now also restores all Frames including \$P_TOOLFRAME.

Calling the **BAxisIncreMotion** macro with Cutter Compensation active no longer causes a motion in the wrong direction for a special case in a specific project file.

The Sin840D DEF command now works correctly when there are more than a space, or a tab, or multiple tabs after the words NCK and/or CHAN.

New option, XYZ_RPY_DIR, is added to the **SetRobotInputType** macro enabling you to build the tool tip position matrix using virtual angles, similar to other options with _DIR key in the name. This case builds a matrix using a different rotation order specific to RPY angles. The matrix is built in the following order: A angle rotates about the Z, then the B angle rotates about the Y and finally the C rotates again about Z axis. The axes are stationary like in other options.

Machine Simulation

Unexpected VERICUT termination no longer occurs after doing a Reset, in a specific project file using machines with a different number of subsystems in each of its 2 setups.

Machine Simulation is enhanced so that non-Stock spinning components are displayed in a Machine view.

Material removal is now correct for a specific project file using a Nakamura mill-turn with a Fanuc18iTB control when the two SubSystems run at the same time.

The collision display is now correct for a specific project file using a ring holder.

Unexpected VERICUT termination no longer occurs after loading a Sin840d.ctl control in a new project and then trying open the Info menu > Sin840D System Frames window.

The VERICUT session no longer locks up during a specific B axis motion.

OptiPath

A new OptiPath setting, Clean-up Volume Tolerance, is added to the OptiPath window: Limits tab enables you to force OptiPath to output a clean-up feedrate for cuts removing material with a volume less than the specified Clean-up Volume Tolerance setting.

Reviewer

Tools in a turret in a reviewer file are now displayed immediately, the same way that they are displayed in VERICUT.

A specific threading insert, in a specific project file, is now oriented correctly when displayed in a Profile view.

The Reviewer can now read STL models in a Reviewer file created from a specific project file that uses STL model files with extremely long folder paths in an encrypted machine file.

The speed of playing a Review file containing circular and helical motions has been improved.

The display of a Reviewer file is now correct in the Workpiece view when the Animation Speed Slider is moved to the right into green Skip Cuts zone.

STEP models now display correctly in all setups in a Review file created from a specific multi-setup project file.

Reviewer files created from a specific project file no longer result in "Error Loading Review Data" errors.

Opening the Setup Plan window with a specific project file loaded, no longer results in a Notice "NullPointerException: String is null" in the logger area.

A new MDI window is added under the Analysis menu in Reviewer. It contains the same Machine Motion Jog features found in the VERICUT MDI window that enable you to incrementally jog the position of the VERICUT machine in the same way that you can jog the real machine. Robot tool axis jogging is not supported.

Tool Manager

Modifying the Neck Diameter on a tap no longer creates a cone instead of a cylinder if the Neck Diameter value is different than the value for the Minor Diameter.

If a Tool Manager instance is associated with a VERICUT project (having been opened by the main VERICUT application) and is currently behind the VERICUT window, and you open Tool Manager again, Tool Manager is now displayed in front of the main VERICUT window.

A Disk Probe Tip type probe tool now correctly retains Position values.

The Add/Modify window for Disk Probe Tips is enhanced or modified as follows:

Feature **Diameter (D)** is renamed **Disk Diameter (D)**.

Feature **Height (H)** is renamed **Disk Height (H)**.

New feature, **Pin Diameter**, is added enabling you to specify the diameter of the disk roller.

New feature, **Pin Offset**, is added enabling you to specify the distance from the centerline of the disk roller to the centerline of the probe tool.

The image that shows what each options represents is updated to include the above changes.

Verification

Saving an IP file and then renaming it in Windows by right click and select Rename no longer results in an 'Invalid in-process file project data' error when you try to open the file in VERICUT.

Using a very large tool library file (.tls) no longer results in degraded VERICUT performance.

The Tool Summary and VERICUT Report are no longer missing 2 of the tools loaded into the 3 straight spindles in a specific project file using the **VcMultiTools** macro.

A problem causing unexpected VERICUT termination in a number of instances with no recognized common actions is fixed.

A Toolbar icon is added for quickly accessing the Favorites window.

Constant Gouge Check is now disabled for polishing tools and fast feed errors are no longer reported for rapid retracts of polishing tools.

The Tool Use Graph now shows all of the tools used in multi-channel sync project files.

Tool Cutting Time and Tool Cutting Distance are now recorded and displayed correctly for a specific project when Cutter Compensation is active.

Opening an In-Process file created from a specific project file no longer outputs label errors that were not in the original simulation.

False Collision errors are no longer output for a specific project file when the Animation Speed Slider is set above 40%.

A specific project file using Siemens Sin840D FRAMES, created in V7.2.1, now simulates the same in V7.3.3 as it did in V7.2.1.

The “Air Time %” value is now displayed correctly in the Status window when using APT_CL files.

Unexpected VERICUT termination no longer occurs when merging a specific In-Process file.

VERICUT now deletes all cutcom_debugxx.sketch files when you exit VERICUT.

The **Save CAS file** option is moved from under File menu to under File menu > Convert.

Unexpected VERICUT termination no longer occurs for a specific project file using a tap tool that has the same Major Diameter, Minor Diameter, and Tip diameter.

The ability to check the fast feed rate against the Fast Feed (FPR) value, when in FPR mode, is added. The Project Tree, Configure Setup menu: Motion tab is modified as follows:

The existing **Fast Feed** label has been changed to "**Fast Feed (FPM)**".

A new field, **Fast Feed (FPR)** is added.

When in FPR mode and the **Fast Feed (FPR)** value is non-zero, the fast feed rate will be checked against the **Fast Feed (FPR)** value.

Otherwise: The fast feed rate will be checked against the **Fast Feed (FPM)** value.

For Tapping and Threading, the fast feed will be checked against the **Fast Feed (FPM)** value.

NOTES:

1. Tapping and Threading (although in FPR mode), will not use the FPR value because this setting is typically much higher than normal FPR settings.
2. The default for the Fast Feed (FPR) is zero. If zero, it will not be used. This allows existing jobs to continue running as is.

Unexpected VERICUT termination no longer occurs when adding a specific Sweep file.

Unexpected VERICUT termination no longer occurs while processing a specific 4 channel Mill-Turn project file.

Unexpected VERICUT termination no longer occurs while processing a specific VERICUT project file.

Miscellaneous

The Siemens 840D feature, \$P_GG[29] X Axis Multiplier, is added to the Library Sin840d control.

The Siemens 840D Cycle800 command now produces the correct angles for a specific project file.

False Overtravel Limit errors are no longer output when executing a G54 command in a specific project file.

The G173 Z move in a specific project file is now correct.

The library hei530 control is enhanced to support tool retracts using the Heidenhain MB MAX command.

Machine Simulation is enhanced to support VTLs using radial tool holder motion.

The **Undo** button now works correctly for the Cut Stock in all setups in a multi-setup project.

A specific project file that uses a specific Siemens 840D control that uses many Siemens DEFINE AS commands now works correctly in V7.3.3 as it did in V7.3.1.

The Siemens Sin840D command \$P_CHBFR[1]=CMIRROR[B] is now supported in the Library Sin840d.ctl.

A new batch command line argument, RECORD_CUTCOND, is added to activate the reviewer and record cutting conditions so that VcPost will report Axial Depth and Radial Width cutting conditions without having the "Record Cutting Conditions" checkbox on the VERICUT "File menu > Properties window: General tab toggled ON (checked).

The **ToolnumToPocket** function now correctly handles cases where the tool list contains a SubSystem ID.

New Macros in V7.3.3

The following new macros are added for V7.3.3

AutosetTableAxisArrayVars

CycleMillPocketRetractOnOff

CycleTurnEndCutX

CycleTurnEndCutZ

EIInterpolateVirtualYZ

Heid_PolarMotionCTP

InterpolationXValue

InterpolationYValue

InterpolationZValue

OkumTurnThread

RadialToolArmLength

RotaryDirShortestDist3

Siemens840DCallSubName

SiemensSetChannelTags

WorkingPlane2AbcVertRiderOffset

WorkingPlane2AbcVertRiderZero

Macros not yet included in the documentation

CutterCompValueWarning

Function — CUTTER COMPENSATION

Status – ACTIVE

Comment — Added V7.3.1

Valid Inputs — Value

0 = No Warning or Error messages output (Default)

1 = messages output as Warnings

2 = messages output as Errors

This macro enables you to specify how you want unreachable motion segments reported.

Set **Override Value** = 0 (Default) to have no Warning messages or Error messages output when unreachable motion segments are encountered.

Set **Override Value** = 1 to have messages output as Warnings when unreachable motion segments are encountered.

Set **Override Value** = 2 to have messages output as Errors when unreachable motion segments are encountered.

This macro should be called after calling the **CutterCompFull** macro set with **Override Value** = 1. This macro is only applicable to cutter compensation full mode.

One of the following messages will be output when unreachable motion segments are encountered:

"Compensation value too large: cannot offset circle with radius = r." or

"Compensation value too large: motion block cannot be reached."

Heid_MotionCT

Function — MOTION

Status — ACTIVE

Comments — Added V7.2.3

Valid Inputs — None

This macro is used to construct a circular motion tangent to the previous motion segment, and the specified new endpoint. The resulting circle direction is determined by its programmed endpoint location and the requirement for a smooth transition from the previous motion.

Heid_PolarMotionCTP

Function — CYLINDRICAL / POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs: None

Processes a spiral command (with optional helix) with inputs specified with Heidenhain specific polar macros. These inputs include radius, angle, and pole (circle center).

The direction of the spiral (CW or CCW) is specified by the direction of the previous move. If the direction of the previous motion is not reasonably tangent with the specified circle, an error will be produced.

NOTE: Since this command uses the radius, angle, and pole from other macros, these other macros must be specified before calling this macro.

See also: **Heid_PolarAngle**, **Heid_PolarIncAngle**, **Heid_PolarRadius**, **Heid_PolarIncRadius**, **HeidIso_PolarAngle**, **HeidIso_PolarRadius**, **Heid_CircleCenterX**, **Heid_CircleCenterY**, **Heid_CircleCenterZ**, **Heid_CircleCenterIncX**, **Heid_CircleCenterIncY**, **Heid_CircleCenterIncZ**

InterpolationXValue

Function — CYLINDRICAL/POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the X Cartesian coordinate value to be used when in Interpolation mode. The Cartesian coordinates will then be converted into machine coordinates based on the current interpolation type. It is intended that this macro will replace **PolarXValue**.

Also see: **InterpolationYValue** and **InterpolationZValue**

InterpolationYValue

Function — CYLINDRICAL/POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the Y Cartesian coordinate value to be used when in Interpolation mode. The Cartesian coordinates will then be converted into machine coordinates based on the current interpolation type. It is intended that this macro will replace **PolarYValue**.

Also see: **InterpolationXValue** and **InterpolationZValue**

InterpolationZValue

Function — CYLINDRICAL/POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the Z Cartesian coordinate value to be used when in Interpolation mode. The Cartesian coordinates will then be converted into machine coordinates based on the current interpolation type. It is intended that this macro will replace **PolarZValue**.

Also see: **InterpolationXValue** and **InterpolationYValue**

Probe

Function — MISCELLANEOUS

Status — ACTIVE

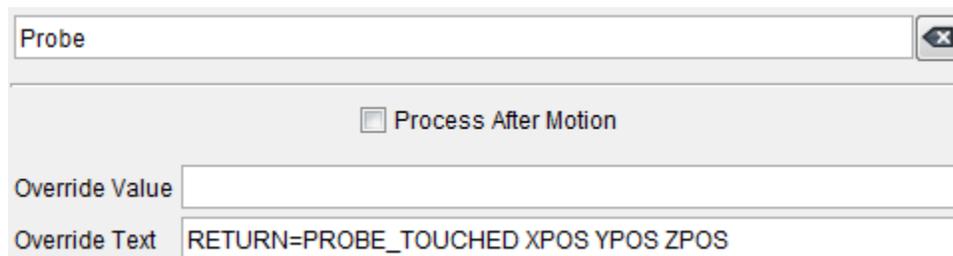
Valid Inputs — Text

This macro turns on probing for the current block. This macro is associated with the active tool component, and the corresponding tool must be a Probe Tool. The axes move until they arrive at the programmed location, or until the probe contacts something. Once contact is made, motion is stopped and the rest of the move is aborted.

If the **Override Text** value begins with "RETURN=", then the string following the equal sign will be interpreted as a variable name in which the return code from the Probe motion will be returned. A return code value of 1 means that a "hit" occurred and a return code value of 0 means that a "hit" did not occur.

If four variable names are supplied then the first variable name will contain the return code, and the next three variable names are used to return the X, Y and Z coordinates of the contact point location in the stock coordinate system. The specified variable names are modal. The variable does not need to be pre-defined. The X, Y and Z positions are only recorded if the probe makes contact.

Example:



The screenshot shows a software interface for configuring a macro. At the top, there is a text box containing the word "Probe" and a small icon to its right. Below this is a checkbox labeled "Process After Motion", which is currently unchecked. Underneath the checkbox are two more text boxes. The first is labeled "Override Value" and is empty. The second is labeled "Override Text" and contains the text "RETURN=PROBE_TOUCHED XPOS YPOS ZPOS".

PROBE_TOUCHED would contain a 1 if the probe made contact and a 0 if it didn't.

XPOS would contain the position value of X.

YPOS would contain the position value of Y.

ZPOS would contain the position value of Z.

NOTES:

1. The values contained in the X, Y, Z variables reflect either the Stock Component coordinate system (default) or the active local coordinate system depending upon the setting of the **MeasureCoordinates** macro.
2. The **Probe** macro requires a CNC Machine Probing license.

Also see: **Measure Coordinates**

RadialToolArmLength

Function — INTERPOLATION

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the arm length this is used with RADIAL TOOL interpolation. In this interpolation mode, the A axis on the tool is rotated to allow for a small movement in the Y direction. The formula is: $Y = \text{SIN}(\text{HEAD ANGLE}) * \text{Arm Length}$.

RadialToolInterpolation

Function — INTERPOLATION

Status — Active

Comment — Added V7.3

Valid Inputs — Text, Value

Text: LINEAR = Apply this interpolation only to non-RAPID moves (default)

ALL = Apply this interpolation to all motions including RAPID

Value: 0 = Turns this interpolation state off (default)

1 = Turns this interpolation state on

This macro turns on and off Radial Tool Interpolation. This interpolation mode was created special for a machine without a Y axis, which used arm length of a right angle tool to create the Y motion.

See also: **RadialToolArmLength**

SetAxisRegisterType

Function — MISCELLANEOUS

Status — Active

Comment — Added V7.3

Valid Inputs — Value

0 = Apply **SetAxisRegisterName** mapping to values and offsets (default)

1 = Apply **SetAxisRegisterName** mapping to only values

This macro changes how the **SetAxisRegisterName** macro is applied. Option 1 was added for the Sin840D GEOAX command.

SetRobotLinearAxisMode

Function — MOTION

Status — ACTIVE

Comments — Added V7.3.1

Valid Inputs — Text, Value

Text:

- ON — activates the linear axis and locks the first rotary at its current position.
- OFF — deactivates the linear axis (locks at its current position) and activates the first rotary axis.
- ON AT — activates the linear axis at a specified position moving the robot to that location.
- MIXED — the robot's linear axis is set as programmed by linear axis Code and the robot's A1 joint is rotated at a calculated angle to achieve the programmed Cartesian XYZ position.
- SEPARATE — the robot's linear axis is not related to the robot's A1 joint and moves using the programmed value from mcd file.

Value: used to specify the linear axis position for this option.

This macro is provides the same functionality provided by the **SetRobotXAxisMode** macro except that it can be used to set more than 1 linear axis.

Example:

To set one axis apply:

```
SetRobotLinearAxisMode OT=FIRST XAXIS ON/OFF/MIXED/SEPARATE AT  
(optional)
```

And to set another one use:

SetRobotLinearAxisMode OT=SECOND YAXIS ON/OFF/MIXED/SEPARATE AT
(optional)

Siemens840DCallSubName

Function — SUBROUTINES

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Text

It is passed the name of the subroutine.

The path name is stripped.

The "_N_" prefix is stripped (if it exists).

The suffix is stripped if found. Suffixes being stripped include: _MPF, _SPF, _TEA, _SEA, _TOA, _UFR, _INI, _GUD, _RPA, _COM, _DEF

The resulting subroutine is then called.

SiemensSetChannelTags

Function — SYNC

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Text

This macro is specific to Siemens. This macro dynamically sets the channel tags. The Sync Method should be set to "Generic - User Defined".

The concept is: the channel tags define which section of the file is to be used for channel 1, and what section of the file is to be used for channel 2.

It will read the SELECT commands, for example

```
SELECT A972804.MPF CH=1
SELECT B972804.MPF CH=2
```

And then strip off any "." extension, and add a "%_N_" prefix, and add a "_MPF" suffix

This macro is processed during the scan pass to automatically set the channel tags (initially, the channel tags can be defaulted to any word that would not be in the file).

Siemens_P_CYCFRAME

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.3.1

Valid Inputs — None

This macro is used when frame variable \$P_CYCFRAME is set/modified. This frame affects the transformation of a point in the NC program to machine coordinates. In Siemens 840D this is just another matrix to apply. Note that \$P_CYCFRAME can be canceled and/or restored using the **SiemensSystemFramesCancel** and **SiemensSystemFramesRestore** macros with **Override Text = CYCFRAME**.

ToolAxisMotionXYZ

Function — MOTION

Status — ACTIVE

Comment — Added V7.3.1

Valid Inputs — Value

This macro supports the Siemens 840D "MOVT=IC" command. It will cause the tool to move the specified distance along the tool axis. This macro only works with a simple XYZ movement (no UVW, no reverse axis, axis must be orthogonal, ...). It does support a rotated local coordinate system.

VERICUT 7.3.3 Interim Release (Composites)

Release Notes

December 9, 2014

VERICUT Version 7.3.3 Composites is available for all supported Windows platforms.

V 7.3.3 contains everything described above for V7.3.2, plus the following additional fixes/enhancements.

VERICUT Composite Simulation (VCS)

Problems Resolved

False "Tape roller is too far from the form at line: ..." errors are no longer output for a specific VCS project file. (441)

Robot motion is now correct for a specific VCS project file. (442, 443)

A false "Spindle Speed is below minimum value" error is no longer output when the Velocity/RPM for the Ply Group is set to a negative value. (450)

VERICUT Composite Programming (VCP)

Enhancements

VCP now accumulates a table of stacked ply names on the "Add material to surfaces" card. (169)

VCP now has Tow area and Weight displayed on the "Machine envelope checks" card. (370)

ATL scrap statistics are now available to a post. (370)

VCP now has the ability to highlight overlaps using the Highlight overlaps option on the "Cumulative tow gap check" card. (370, 439)

The "Points" sub-card of the "Geometry Construction" card now has additional buttons enabling you to specify how to project a point to the form's surface. Projection can be perpendicular to the form's surface, or along one of the active coordinate system's axes. (375)

The Drop-down menu for the table of surface names on "Import form, mold or mandrel" card now has a choice to "Delete selected surface". (379)

VCP now allows a post to determine the robot joint angles for any commanded end-effector location. There is also a reserved post array (DHPOse) which can be used to define an initial pose. (388)

VCP now displays, on the "Rule compliance check" card, a count of the number of "x"s where there are ply angle deviations, steering issues or compression failures. Each of these counts is displayed between the button (**Highlight ply angle deviations**, **Highlight steering radius violations** and **Highlight inadequate roller compression**) that triggered creation of the "x"s and the **Erase** button which will remove them. (418)

VCP now scales the size of icons When the Text "Smaller" and "Larger" buttons on the "About VCP" card are selected. The sizes of the icons are scaled to match the new text size. The text size will change immediately. But the icon sizes WILL NOT CHANGE until you exit VCP and re-enter. (423)

VCP is enhanced so that the "Post-processor options" card now enables you to specify multiple Post-processor options at the same location. (433)

VCP is enhanced to enable you to cancel a Zoom operation before selecting the second point defining the zoom box. Drag the mouse so that the size of the zoom box is very small (8 pixels or less in both directions) and then release the mouse button. The zoom box will go away and the zoom operation will be cancelled. (434)

VCP is enhanced to add the text from the ply identifier field to the bitmap image created during post-processing. (449)

VCP is enhanced to enable dragging the separation between the tabbed area and the graphics area -in order to make the graphics region narrower and the tab region wider. (453)

Problems Resolved

In a specific project file, the -45 degree courses are now laid correctly. (428)

When the uni-directional option on the "Tow-packet links" card is toggled on, spinning for a zero-orientation (paths parallel to the rotation axis) no longer causes the VCP session to lock-up. (429)

False "Programmed position out of reach" errors are no longer output for a specific robot VCS program. (443)

CAM Interfaces

CATIA-to-VERICUT Interface (CATV)

Unexpected VERICUT and Tool Manager termination no longer occurs when certain tools created from CATIA User Representations are passed into the VERICUT tool library.

The CATIA-to-VERICUT Interface now correctly passes a specific tool that has a driven point (corrector in CATIA terminology) with a blank "type" to VERICUT.

The CATV.exe now looks for the CGTECH_SINGLE_PLATFORM environment variable and automatically inserts "Limited" in the command line it uses to invoke VERICUT.

CATIA-to-VERICUT Drilling and Fastening Interface (CAT2VDAF)

The CATIA-to-VERICUT Drilling and Fastening Interface is enhanced to enable mirroring skin, structure or fixture models in the same way as the CATIA-to-VERICUT Interface.

The CATIA-to-VERICUT Interface is enhanced to look for milling tool profile sketches in both part bodies and in geometrical sets when passing 3D-tools from CATIA through the Custom Tool Geometry list in CATV.

The CAT2VDAF.exe now looks for the CGTECH_SINGLE_PLATFORM environment variable and automatically inserts "Limited" in the command line it uses to invoke VERICUT.

Edgecam-to-VERICUT Interface

The Edgecam-to-VERICUT Interface is enhanced to support multiple G-Code offsets Edgecam 2014 R1 and later.

The Edgecam-to-VERICUT Interface is enhanced to support selecting from multiple attach components for Edgecam 2014 R1 and later.

Deleting sequences in the EdgeCAM-to-VERICUT interface now works correctly in the latest EdgeCAM-to-VERICUT interface for Edgecam2014 R2.

Using a horizontal machine and using offsets within the EdgeCAM interface, the coordinates now travel with the stock correctly in the Edgecam-to-VERICUT Interface for Edgecam 2014 R2.

GibbsCAM-to-VERICUT Interface (GibbsV)

The GibbsCAM-to-VERICUT Interface is enhanced so that when using the GibbsV to post programs when selecting Generate and Run and the post-processor has an option in it that controls the extension of the output file, GibbsV will read this field from the post-processor and if there is an extension present, the file will be output with that extension. If the field is left blank, then the file is output with no extension.

The GibbsCAM-to-VERICUT Interface now correctly saves the selected Attach Axis System when closing the GibbsV window and then reloads it when the GibbsV window is reopened.

The GibbsCAM-to-VERICUT Interface now transfers all GibbsCAM coordinate systems to VERICUT.

The GibbsCAM-to-VERICUT Interface now correctly transfers 2 specific tap tools, in a specific GibbsCAM VNC file, to VERICUT.

Mastercam-to-VERICUT Interface (MCAMV)

The Mastercam-to-VERICUT Interface now transfers a specific coordinate system from Mastercam into the correct location in VERICUT.

The Mastercam-to-VERICUT Interface now transfers a specific Lathe tool insert with the insert thickness in the correct direction.

The Mastercam-to-VERICUT Interface now transfers a specific Custom tool to VERICUT with the correct units.

The Mastercam-to-VERICUT Interface now transfers a specific lathe tool assembly, consisting of metric tool insert and an inch holder, correctly to VERICUT.

The Mastercam-to-VERICUT Interface now transfers a specific Mastercam tap tool assembly correctly to VERICUT.

The Mastercam-to-VERICUT Interface is enhanced to include a Calculate Relative to Location option in the VERICUT G-Code Tables section of the MCAMV VERICUT Interface window. This feature enables you to have a relational offset recalculated in the machine position where the offset will be used. The new position is immediately calculated and stored and therefore is not dependent on the machine position when the offset is activated.

The VERICUT icon in Mastercam used to start the Mastercam-to-VERICUT Interface now displays correctly when switching to “Large Buttons” using Mastercam’s Settings > Configuration > Screen > Large Buttons checkbox.

The Stock Delta X, Y, and Z text fields, in the MCAMV Options window, have been lengthened from 36 to 60 pixels enabling you to see all of the characters entered in the text boxes.

G-Code Processing

New macro, **AutosetTableAxisArrayVars**, is added to provide support for Okuma OSP BiDirectional Alpha Array Work Offset System Variables.

Offsets are now correctly processed for a specific project file using a Siemens 840D SUPA command.

New macro, **SiemensSetChannelTags**, is added to support a new Siemens 840D syncing method.

Support is added for Fast Feed IPR values. The existing "Fast Feed" label on the Project Tree Configure Setup; Motion tab is changed to "Fast Feed (FPM)". A new field "Fast Feed (FPR)" is added. If in FPR mode and the Fast Feed (FPR) is non-zero, fast feed will be checked against the Fast Feed (FPR) value. Otherwise: The fast feed will be checked against the Fast Feed (FPM) value. For Tapping and Threading, the fast feed will be checked against the Fast Feed (FPM) value.

Macro **HelicalAddFullLoops** now produces correct number of turns with the correct pitch.

New macro, **Siemens840DCallSubName**, is added to enable removing the path and suffixes from a Siemens EXTCALL command.

Adding **CutterCompFull** Override Value = 1 to Events, Start of Processing of a specific project file, no longer causes VERICUT to unexpectedly jump into subroutines that are not called by the NC program.

Using **CutterCompFull** Override Value = 1 with a specific tool in a specific project file no longer results in incorrect motion.

Using Okuma OSP G72 CycleTurnRoughFace now correctly outputs a Fast feed for the G00 on the final pass in a specific project file.

A new option, **Shortest Distance 3** is added to the Absolute Rotary Direction feature on the Configuration menu > Control settings window > Rotary tab. It functions the same way as the **Shortest Distance** option, but no error messages are output when a 180 degree rotary move is programmed.

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

False "Invalid circle statement ..." messages are no longer output for a specific project file using macros **RpcpOn** and **RpctInput** with Override Value = 1.

When a G96 CSS is active on the part spindle while processing a CYCLE84 tapping cycle on the tool spindle no longer results in false "Error: Feed per rev of "2.1856" does not match pitch of tap tool "13" at line: ..." messages.

Unexpected VERICUT termination no longer occurs while processing a specific project when doing merged turning on a machine with an index chuck.

Processing **IjkAAxisMotion**, **IjkBAxisMotion**, and **IjkCAxisMotion** macros now produces the correct results in a specific project file also using a **CoupleAxisOn** macro.

The macro **SiemensSystemFramesCancel** now cancels all Frames including \$P_TOOLFRAME. The **SiemensSystemFramesRestore** macro now also restores all Frames including \$P_TOOLFRAME.

Calling the **BAxisIncreMotion** macro with Cutter Compensation active no longer causes a motion in the wrong direction for a special case in a specific project file.

The Sin840D DEF command now works correctly when there are more than a space, or a tab, or multiple tabs after the words NCK and/or CHAN.

New option, XYZ_RPY_DIR, is added to the **SetRobotInputType** macro enabling you to build the tool tip position matrix using virtual angles, similar to other options with _DIR key in the name. This case builds a matrix using a different rotation order specific to RPY angles. The matrix is built in the following order: A angle rotates about the Z, then the B angle rotates about the Y and finally the C rotates again about Z axis. The axes are stationary like in other options.

Machine Simulation

Unexpected VERICUT termination no longer occurs after doing a Reset, in a specific project file using machines with a different number of subsystems in each of its 2 setups.

Machine Simulation is enhanced so that non-Stock spinning components are displayed in a Machine view.

Material removal is now correct for a specific project file using a Nakamura mill-turn with a Fanuc18iTB control when the two SubSystems run at the same time.

The collision display is now correct for a specific project file using a ring holder.

Unexpected VERICUT termination no longer occurs after loading a Sin840d.ctl control in a new project and then trying open the Info menu > Sin840D System Frames window.

The VERICUT session no longer locks up during a specific B axis motion.

OptiPath

A new OptiPath setting, Clean-up Volume Tolerance, is added to the OptiPath window: Limits tab enables you to force OptiPath to output a clean-up feedrate for cuts removing material with a volume less than the specified Clean-up Volume Tolerance setting.

Reviewer

Tools in a turret in a reviewer file are now displayed immediately, the same way that they are displayed in VERICUT.

A specific threading insert, in a specific project file, is now oriented correctly when displayed in a Profile view.

The Reviewer can now read STL models in a Reviewer file created from a specific project file that uses STL model files with extremely long folder paths in an encrypted machine file.

The speed of playing a Review file containing circular and helical motions has been improved.

The display of a Reviewer file is now correct in the Workpiece view when the Animation Speed Slider is moved to the right into green Skip Cuts zone.

STEP models now display correctly in all setups in a Review file created from a specific multi-setup project file.

Reviewer files created from a specific project file no longer result in "Error Loading Review Data" errors.

Opening the Setup Plan window with a specific project file loaded, no longer results in a Notice "NullPointerException: String is null" in the logger area.

A new MDI window is added under the Analysis menu in Reviewer. It contains the same Machine Motion Jog features found in the VERICUT MDI window that enable you to incrementally jog the position of the VERICUT machine in the same way that you can jog the real machine. Robot tool axis jogging is not supported.

Tool Manager

Modifying the Neck Diameter on a tap no longer creates a cone instead of a cylinder if the Neck Diameter value is different than the value for the Minor Diameter.

If a Tool Manager instance is associated with a VERICUT project (having been opened by the main VERICUT application) and is currently behind the VERICUT window, and you open Tool Manager again, Tool Manager is now displayed in front of the main VERICUT window.

A Disk Probe Tip type probe tool now correctly retains Position values.

The Add/Modify window for Disk Probe Tips is enhanced or modified as follows:

Feature **Diameter (D)** is renamed **Disk Diameter (D)**.

Feature **Height (H)** is renamed **Disk Height (H)**.

New feature, **Pin Diameter**, is added enabling you to specify the diameter of the disk roller.

New feature, **Pin Offset**, is added enabling you to specify the distance from the centerline of the disk roller to the centerline of the probe tool.

The image that shows what each options represents is updated to include the above changes.

Verification

Saving an IP file and then renaming it in Windows by right click and select Rename no longer results in an 'Invalid in-process file project data' error when you try to open the file in VERICUT.

Using a very large tool library file (.tls) no longer results in degraded VERICUT performance.

The Tool Summary and VERICUT Report are no longer missing 2 of the tools loaded into the 3 straight spindles in a specific project file using the **VcMultiTools** macro.

A problem causing unexpected VERICUT termination in a number of instances with no recognized common actions is fixed.

A Toolbar icon is added for quickly accessing the Favorites window.

Constant Gouge Check is now disabled for polishing tools and fast feed errors are no longer reported for rapid retracts of polishing tools.

The Tool Use Graph now shows all of the tools used in multi-channel sync project files.

Tool Cutting Time and Tool Cutting Distance are now recorded and displayed correctly for a specific project when Cutter Compensation is active.

Opening an In-Process file created from a specific project file no longer outputs label errors that were not in the original simulation.

False Collision errors are no longer output for a specific project file when the Animation Speed Slider is set above 40%.

A specific project file using Siemens Sin840D FRAMES, created in V7.2.1, now simulates the same in V7.3.3 as it did in V7.2.1.

The "Air Time %" value is now displayed correctly in the Status window when using APT_CL files.

Unexpected VERICUT termination no longer occurs when merging a specific In-Process file.

VERICUT now deletes all cutcom_debugxx.sketch files when you exit VERICUT.

The **Save CAS file** option is moved from under File menu to under File menu > Convert.

Unexpected VERICUT termination no longer occurs for a specific project file using a tap tool that has the same Major Diameter, Minor Diameter, and Tip diameter.

The ability to check the fast feed rate against the Fast Feed (FPR) value, when in FPR mode, is added. The Project Tree, Configure Setup menu: Motion tab is modified as follows:

The existing **Fast Feed** label has been changed to "**Fast Feed (FPM)**".

A new field, **Fast Feed (FPR)** is added.

When in FPR mode and the **Fast Feed (FPR)** value is non-zero, the fast feed rate will be checked against the **Fast Feed (FPR)** value.

Otherwise: The fast feed rate will be checked against the **Fast Feed (FPM)** value.

For Tapping and Threading, the fast feed will be checked against the **Fast Feed (FPM)** value.

NOTES:

3. Tapping and Threading (although in FPR mode), will not use the FPR value because this setting is typically much higher than normal FPR settings.
4. The default for the Fast Feed (FPR) is zero. If zero, it will not be used. This allows existing jobs to continue running as is.

Unexpected VERICUT termination no longer occurs when adding a specific Sweep file.

Unexpected VERICUT termination no longer occurs while processing a specific 4 channel Mill-Turn project file.

Unexpected VERICUT termination no longer occurs while processing a specific VERICUT project file.

Miscellaneous

The Siemens 840D feature, \$P_GG[29] X Axis Multiplier, is added to the Library Sin840d control.

The Siemens 840D Cycle800 command now produces the correct angles for a specific project file.

False Overtravel Limit errors are no longer output when executing a G54 command in a specific project file.

The G173 Z move in a specific project file is now correct.

The library hei530 control is enhanced to support tool retracts using the Heidenhain MB MAX command.

Machine Simulation is enhanced to support VTLs using radial tool holder motion.

The **Undo** button now works correctly for the Cut Stock in all setups in a multi-setup project.

A specific project file that uses a specific Siemens 840D control that uses many Siemens DEFINE AS commands now works correctly in V7.3.3 as it did in V7.3.1.

The Siemens Sin840D command \$P_CHBFR[1]=CMIRROR[B] is now supported in the Library Sin840d.ctl.

A new batch command line argument, RECORD_CUTCOND, is added to activate the reviewer and record cutting conditions so that VcPost will report Axial Depth and Radial Width cutting conditions without having the "Record Cutting Conditions" checkbox on the VERICUT "File menu > Properties window: General tab toggled ON (checked).

The **ToolnumToPocket** function now correctly handles cases where the tool list contains a SubSystem ID.

New Macros in V7.3.3

The following new macros are added for V7.3.3

AutosetTableAxisArrayVars

CycleMillPocketRetractOnOff

CycleTurnEndCutX

CycleTurnEndCutZ

EIInterpolateVirtualYZ

Heid_PolarMotionCTP

InterpolationXValue

InterpolationYValue

InterpolationZValue

OkumTurnThread

RadialToolArmLength

RotaryDirShortestDist3

Siemens840DCallSubName

SiemensSetChannelTags

WorkingPlane2AbcVertRiderOffset

WorkingPlane2AbcVertRiderZero

Macros not yet included in the documentation

CutterCompValueWarning

Function — CUTTER COMPENSATION

Status – ACTIVE

Comment — Added V7.3.1

Valid Inputs — Value

0 = No Warning or Error messages output (Default)

1 = messages output as Warnings

2 = messages output as Errors

This macro enables you to specify how you want unreachable motion segments reported.

Set **Override Value** = 0 (Default) to have no Warning messages or Error messages output when unreachable motion segments are encountered.

Set **Override Value** = 1 to have messages output as Warnings when unreachable motion segments are encountered.

Set **Override Value** = 2 to have messages output as Errors when unreachable motion segments are encountered.

This macro should be called after calling the **CutterCompFull** macro set with **Override Value** = 1. This macro is only applicable to cutter compensation full mode.

One of the following messages will be output when unreachable motion segments are encountered:

"Compensation value too large: cannot offset circle with radius = r." or

"Compensation value too large: motion block cannot be reached."

Heid_MotionCT

Function — MOTION

Status — ACTIVE

Comments — Added V7.2.3

Valid Inputs — None

This macro is used to construct a circular motion tangent to the previous motion segment, and the specified new endpoint. The resulting circle direction is determined by its programmed endpoint location and the requirement for a smooth transition from the previous motion.

Heid_PolarMotionCTP

Function — CYLINDRICAL / POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs: None

Processes a spiral command (with optional helix) with inputs specified with Heidenhain specific polar macros. These inputs include radius, angle, and pole (circle center).

The direction of the spiral (CW or CCW) is specified by the direction of the previous move. If the direction of the previous motion is not reasonably tangent with the specified circle, an error will be produced.

NOTE: Since this command uses the radius, angle, and pole from other macros, these other macros must be specified before calling this macro.

See also: **Heid_PolarAngle**, **Heid_PolarIncAngle**, **Heid_PolarRadius**, **Heid_PolarIncRadius**, **HeidIso_PolarAngle**, **HeidIso_PolarRadius**, **Heid_CircleCenterX**, **Heid_CircleCenterY**, **Heid_CircleCenterZ**, **Heid_CircleCenterIncX**, **Heid_CircleCenterIncY**, **Heid_CircleCenterIncZ**

InterpolationXValue

Function — CYLINDRICAL/POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the X Cartesian coordinate value to be used when in Interpolation mode. The Cartesian coordinates will then be converted into machine coordinates based on the current interpolation type. It is intended that this macro will replace **PolarXValue**.

Also see: **InterpolationYValue** and **InterpolationZValue**

InterpolationYValue

Function — CYLINDRICAL/POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the Y Cartesian coordinate value to be used when in Interpolation mode. The Cartesian coordinates will then be converted into machine coordinates based on the current interpolation type. It is intended that this macro will replace **PolarYValue**.

Also see: **InterpolationXValue** and **InterpolationZValue**

InterpolationZValue

Function — CYLINDRICAL/POLAR

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the Z Cartesian coordinate value to be used when in Interpolation mode. The Cartesian coordinates will then be converted into machine coordinates based on the current interpolation type. It is intended that this macro will replace **PolarZValue**.

Also see: **InterpolationXValue** and **InterpolationYValue**

Probe

Function — MISCELLANEOUS

Status — ACTIVE

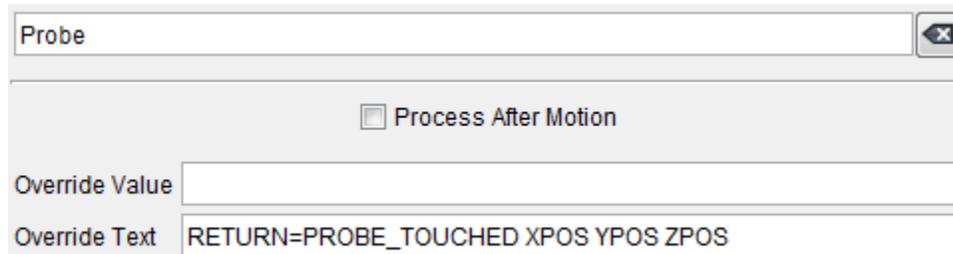
Valid Inputs — Text

This macro turns on probing for the current block. This macro is associated with the active tool component, and the corresponding tool must be a Probe Tool. The axes move until they arrive at the programmed location, or until the probe contacts something. Once contact is made, motion is stopped and the rest of the move is aborted.

If the **Override Text** value begins with "RETURN=", then the string following the equal sign will be interpreted as a variable name in which the return code from the Probe motion will be returned. A return code value of 1 means that a "hit" occurred and a return code value of 0 means that a "hit" did not occur.

If four variable names are supplied then the first variable name will contain the return code, and the next three variable names are used to return the X, Y and Z coordinates of the contact point location in the stock coordinate system. The specified variable names are modal. The variable does not need to be pre-defined. The X, Y and Z positions are only recorded if the probe makes contact.

Example:



Probe	
<input type="checkbox"/> Process After Motion	
Override Value	
Override Text	RETURN=PROBE_TOUCHED XPOS YPOS ZPOS

PROBE_TOUCHED would contain a 1 if the probe made contact and a 0 if it didn't.

XPOS would contain the position value of X.

YPOS would contain the position value of Y.

ZPOS would contain the position value of Z.

NOTES:

3. The values contained in the X, Y, Z variables reflect either the Stock Component coordinate system (default) or the active local coordinate system depending upon the setting of the **MeasureCoordinates** macro.
4. The **Probe** macro requires a CNC Machine Probing license.

Also see: **Measure Coordinates**

RadialToolArmLength

Function — INTERPOLATION

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Value

This macro sets the arm length this is used with RADIAL TOOL interpolation. In this interpolation mode, the A axis on the tool is rotated to allow for a small movement in the Y direction. The formula is: $Y = \sin(\text{HEAD ANGLE}) * \text{Arm Length}$.

RadialToolInterpolation

Function — INTERPOLATION

Status — Active

Comment — Added V7.3

Valid Inputs — Text, Value

Text: LINEAR = Apply this interpolation only to non-RAPID moves (default)

ALL = Apply this interpolation to all motions including RAPID

Value: 0 = Turns this interpolation state off (default)

1 = Turns this interpolation state on

This macro turns on and off Radial Tool Interpolation. This interpolation mode was created special for a machine without a Y axis, which used arm length of a right angle tool to create the Y motion.

See also: **RadialToolArmLength**

SetAxisRegisterType

Function — MISCELLANEOUS

Status — Active

Comment — Added V7.3

Valid Inputs — Value

0 = Apply **SetAxisRegisterName** mapping to values and offsets (default)

1 = Apply **SetAxisRegisterName** mapping to only values

This macro changes how the **SetAxisRegisterName** macro is applied. Option 1 was added for the Sin840D GEOAX command.

SetRobotLinearAxisMode

Function — MOTION

Status — ACTIVE

Comments — Added V7.3.1

Valid Inputs — Text, Value

Text:

- ON — activates the linear axis and locks the first rotary at its current position.
- OFF — deactivates the linear axis (locks at its current position) and activates the first rotary axis.
- ON AT — activates the linear axis at a specified position moving the robot to that location.
- MIXED — the robot's linear axis is set as programmed by linear axis Code and the robot's A1 joint is rotated at a calculated angle to achieve the programmed Cartesian XYZ position.
- SEPARATE — the robot's linear axis is not related to the robot's A1 joint and moves using the programmed value from mcd file.

Value: used to specify the linear axis position for this option.

This macro is provides the same functionality provided by the **SetRobotXAxisMode** macro except that it can be used to set more than 1 linear axis.

Example:

To set one axis apply:

```
SetRobotLinearAxisMode OT=FIRST XAXIS ON/OFF/MIXED/SEPARATE AT  
(optional)
```

And to set another one use:

SetRobotLinearAxisMode OT=SECOND YAXIS ON/OFF/MIXED/SEPARATE AT
(optional)

Siemens840DCallSubName

Function — SUBROUTINES

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Text

It is passed the name of the subroutine.

The path name is stripped.

The "_N_" prefix is stripped (if it exists).

The suffix is stripped if found. Suffixes being stripped include: _MPF, _SPF, _TEA, _SEA, _TOA, _UFR, _INI, _GUD, _RPA, _COM, _DEF

The resulting subroutine is then called.

SiemensSetChannelTags

Function — SYNC

Status — ACTIVE

Comment — Added V7.3.3

Valid Inputs — Text

This macro is specific to Siemens. This macro dynamically sets the channel tags. The Sync Method should be set to "Generic - User Defined".

The concept is: the channel tags define which section of the file is to be used for channel 1, and what section of the file is to be used for channel 2.

It will read the SELECT commands, for example

```
SELECT A972804.MPF CH=1  
SELECT B972804.MPF CH=2
```

And then strip off any "." extension, and add a "%_N_" prefix, and add a "_MPF" suffix

This macro is processed during the scan pass to automatically set the channel tags (initially, the channel tags can be defaulted to any word that would not be in the file).

Siemens_P_CYCFRAME

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.3.1

Valid Inputs — None

This macro is used when frame variable \$P_CYCFRAME is set/modified. This frame affects the transformation of a point in the NC program to machine coordinates. In Siemens 840D this is just another matrix to apply. Note that \$P_CYCFRAME can be canceled and/or restored using the **SiemensSystemFramesCancel** and **SiemensSystemFramesRestore** macros with **Override Text = CYCFRAME**.

ToolAxisMotionXYZ

Function — MOTION

Status — ACTIVE

Comment — Added V7.3.1

Valid Inputs — Value

This macro supports the Siemens 840D "MOVT=IC" command. It will cause the tool to move the specified distance along the tool axis. This macro only works with a simple XYZ movement (no UVW, no reverse axis, axis must be orthogonal, ...). It does support a rotated local coordinate system.

VERICUT 7.3.4 Interim Release

Release Notes

January 22, 2015

VERICUT Version 7.3.4 is available for all supported Windows platforms. V 7.3.4 contains everything described above for V7.3.3, plus the following additional fixes/enhancements.

CAM Interfaces

CATIA-to-VERICUT Interface (CATV)

Models passed through CATV to VERICUT are no longer stretched or scaled along the X-axis when the model is defined with other than a unit vector.

CATV is enhanced to pass right hand (spindle direction=CW) and left hand (spindle direction=CCW) tool information created in CATIA to VERICUT.

CATV now passes a specific tool from CATIA to VERICUT in the correct orientation. (13695)

CATIA-to-VERICUT Drilling and Fastening Interface (CAT2VDAF)

T-Slot cutters are now passed correctly through CATV from CATIA to VERICUT.

EdgeCAM-to-VERICUT Interface (ECV)

A network path that uses a \\ in the path is now handled correctly by ECV.

Mastercam-to-VERICUT Interface (MCAMV)

MCAMV now correctly passes a specific tap and a specific counterboring (shoulder) drill from Mastercam to VERICUT.

MCAMV now correctly passes a specific tapered ball end mill from Mastercam to VERICUT.

NX-to-VERICUT Interface (NXV)

NXV is enhanced to support multiple Work Offsets.

Turning tool holders, using 55 degree diamond turning inserts, now get passed correctly from NX through NXV to VERICUT.

NXV now correctly passes “tool solid representations” from NX to VERICUT.

NXV is enhanced to pass Mill tool driven point data from NX to VERICUT.

NXV now correctly passes 3D tools defined as solid bodies, imported from TDM to NX, to VERICUT.

Pro/Engineer-to-VERICUT Interface (PROEV)

Fixture components are now passed correctly from CREO 2 through PROEV to VERICUT.

AUTO-DIFF

In the AUTO-DIFF window: Settings Tab, when Comparison Method is set to Surface the "Minimum Excess" check box in the Comparison Tolerance section no longer disappears when in Text Larger mode.

A false gouge error, "Error: Maximum Allowable Gouge violation for tool ...", is no longer output when processing a specific project file with Constant Gouge Check turned on.

G-Code Processing

New option, CurToolVectorLocal, is added to the **SetDynamicVars** macro to enable populating three variables with the I, J, K values of the tool axis vector in the local Driven Point Zero.

False “Invalid circle ...” errors are no longer output with the **CutterCompFull** macro, Override Value = 0 for a specific project file. The motion for the same project file is also now correct when the **CutterCompFull** macro, Override Value = 1 for the same situation.

A specific project file that worked correctly when processing an indirect value used in an Override Value in V7.3.1, but failed in V7.3.3, now works correctly again in V7.3.4.

CycleMillPocket macro values are now correctly initialized / reset.

Rotary probing and touching now work correctly in incremental mode.

The **CycleTurnNextLine** macro is enhanced so that it is applied regardless of the cycle type.

New macro **FastenerSetTotalThickness** enables you to set the total stack thickness at the current location.

A specific project file that worked correctly when processing a CC function (Circle Center with Incremental Polar Angle) in V7.3.1, but failed in V7.3.3, now works correctly again in V7.3.4.

The G71 Turning Cycle in a specific project file now correctly removes material parallel to Z Axis.

When processing a Siemens EXECUTE command in a specific project file, the XAxisMultiplier no longer gets reversed when creating the table in CYCLE95.

The Siemens 840d ATRANS C command now works correctly when called more than 1 time.

New function, **IncrementalValue**, is added to provide an alternate way of handling the Siemens IC command. This function sets a flag stating that the value is incremental. It causes any Override Value to be ignored for this value. In the Word Address table, the Conditional State “ABS_INC” has been enhanced to support either Incremental mode, or a value set with the **IncrementalValue** function. A few specific macros have been enhanced to look to see if this new incremental flag is set. These macros include:

XAxisMotion	CAxisMotion	B2AxisMotion
YAxisMotion	UAxisMotion	C2AxisMotion
ZAxisMotion	VAxisMotion	CircleCenterX
AAxisMotion	WAxisMotion	CircleCenterY
BAxisMotion	A2AxisMotion	CircleCenterZ

New macros **SiemensPAROTVersion** and **SiemensTCOFR** are added to support the Siemens PAROT/TCOFR command.

New macro, **RadiusDeltaPerRev**, to is added to support a spiral motion using the Q register for a pitch value indicating the radius increase or decrease per loop.

If the specified radius delta value does not match the other spiral parameters (start and end radii, and the angle difference between start and end points) a warning is output.

In the case of a mismatch a valid spiral motion is output with the radius delta value adjusted to match the spiral parameters to produce the best matching number of full loops.

Unexpected VERICUT termination no longer occurs when you highlight a blank line in the NC Program window.

Unexpected VERICUT termination no longer occurs in a specific project file using the **CutterCompFull** macro with an Override Value = 1.

Machine Simulation

A CYCLE800 command now drills holes in the correct location for a specific project file.

Link component connections are no longer lost after closing a specific machine file using German characters in the component names and then re-opening it.

The left Tricept head, in a specific project file using sync motion, now positions correctly.

The part-off cut-stock no longer gets hidden inside the sub spindle unit, in a specific project file, when stepping through simulation with the Animation Speed Slider in the Skip Motions range.

“Rapid into stock” errors are now reported correctly when using the **SetVcMultiTools** macro with an Override Value = 1.

Reviewer

The VERICUT Drill and fastener Simulation Reviewer is enhanced to include 2 new options in the Info menu. **Detailed Report** displays the Fastener Locations window. **Summary** displays the Simulated Location Summary window.

The Reviewer is enhanced with new option PDF in the File menu enabling you to view an embedded report file like in the iPad Reviewer App. This option is only enabled when the review file is saved with a PDF report file.

The Reviewer is enhanced with new option MDI in the Analysis menu enabling you to move machine axes interactively. This feature is available in both the Windows Reviewer and in the iPad Reviewer App.

The iPad Reviewer App is enhanced to include the Snap to Orthogonal View feature.

The Reviewer no longer displays X-Caliper window results in English when operating in a Japanese computer environment.

VERICUT Drill and Fastener

The Auto-set working directory to current project folder option in the Preferences window: Start-up tab now works correctly the first time that VERICUT Drill and Fastener Simulation is launched from a desktop icon.

New feature, **Merge Locations**, is added to VERICUT Drill and Fastener Programming enabling you to compare and merge Locations files.

Verification

False collision errors are no longer output for a specific project file when the Animation Speed Slider is set to 100%.

Images saved using the AutoSave window, View Capture "Text" option now display correctly to a Report View Capture table and Shade Copy table.

VERICUT Report is enhanced to enable adding AutoSave, Auto Error images to a Report View Capture table.

Inspection Measurement and Datum symbols are now displayed correctly in PDF Inspection reports.

The Send Control Settings Messages setting in the Output Options window: Messages tab is now correctly saved in the project file.

The VERICUT session no longer hangs when processing a specific project file.

VERICUT now recognizes APT commands SPINDL/ON,CW and SPINDL/ON,CCW and the spindle direction based on these commands are displayed correctly in the Status window.

Unexpected VERICUT termination no longer occurs when using X-Caliber to measure a tapped hole in a specific VERICUT Solid File model (.vct).

Strange error messages “Error: *** DANGER *** Action Queue has Invalid Current Class” and “Error: CutCom Off request when last motion was not processed” are no longer output when processing a specific project file.

The VERICUT session no longer takes an extremely long time to process a specific block in a specific project file.

VERICUT is enhanced to support a B rotary that is specified as: Absolute/RotaryDirPosCCW (both in the GUI and in Start of Processing) with Orivect motion.

Section window “Pl Type” and "Color" option lists now have the drop-down arrow indicator always visible.

An NC Program Review manually defined Start Point now displays in the correct position when subroutines are located in the main program file.

A specific project file now produces the correct motion when the **CutterCompFull** macro is used with an Override Value = 1.

The VERICUT session no longer hangs when processing a specific project file with both Calculate Min. Cutter Extension active and a Near Miss value set for Visible Fixture and Active Tools' Cutters.

VERICUT is enhanced to recognize label TAGs if they contain a dash "-" in their names.

The VERICUT-TOOLID comment record is now processed correctly by VERICUT.

VERICUT now stops correctly for Stop At conditions after doing a Merge IP File.

X-Caliper now displays the correct thread direction for an outside turning thread in a specific project file.

Unexpected VERICUT termination no longer occurs when opening a specific project file that uses a FRAME statement with an invalid syntax.

The Cut Stock image quality in the Workpiece view is enhanced.

Miscellaneous

A specific STEP file is now read into VERICUT the same regardless of the session units (Inch or Millimeter).

Czech characters are now displayed correctly in V7.3.4.

The library Sin840d.ctl is enhanced to use the latest method for TRANS & ATRANS for G-Code Processing, Registers, A, B, C, U, V & W with conditional G 58-59. This brings it into line as per G-Code Processing, Registers, X *, Y * & Z *. G-Code Processing, Registers, X * with GEOAX_1_U = 1 & TRANS is now correct.

A problem causing licenses to be lost after using the Import CAD Tool window to import a STEP file on computers set to a non-English local environment for languages that use a "," for a decimal place symbol instead of a ".".

The NC Program window now gets refreshed after it is edited using a third party editor.

VERICUT Report is enhanced to enable adding AutoSave, Auto Error images to a Report View Capture table.

Importing tools from the Zoller Database is enhanced to import the description text from the Zoller Database to the tool library in Tool Manager.

Changing the visibility of tools like taps, probes and knife tool types that have a cutting portion and a non-cutting portion now affects the whole tool and not just the cutting portion.

Probe tool colors in a tool library now have the same colors when referenced in another tool library.

A center drill defined using an NX CLS "TLDATA/DRILL,MILL, ..." statement is now supported in VERICUT.

The library sin840d control is updated to use so \$P_F is REAL and F * uses \$P_F. FB and FAD are also updated to use \$P_F.

New Macros in V7.3.4

The following new macros are added for V7.3.4

EI_SpinningMotionCheck

FastenerSetTotalThickness

SiemensPAROT

SiemensPAROTVersion

SiemensTCOFR

ToolOffsetActiveToolOnOff

Macros not yet included in the documentation

SiemensPAROTVersion

Function — ROTATION_PLANE

Status — Active

Comment — V7.3.4

Valid Inputs — Value

This macro is specific to the Siemens 840D PAROT command and the CYCLE800 subroutine (Working Plane programming). Its purpose is to switch between the new PAROT/TCOFR logic and the old PAROT logic.

When Override Value=0 (default), the existing PAROT logic is applied to solve Working Plane rotary axis and \$P_PARTFRAME frame and its matrix.

When Override Value=1, the new logic, similar to the Siemens 840D control, is used. With this logic, calculations are performed in two steps. The first step is the calculation of rotary angles using the TCOFR command (see **SiemensTCOFR** macro) where 2 sets of angles are calculated and saved in a system variable processed by CYCLE800 subroutine where one of the sets is selected as the valid solution. The second step is the Siemens PAROT command (see **SiemensPAROT** macro) where using the specified rotary angles the \$P_PARTFRAME is calculated and activated. The new logic doesn't solve the problem of which rotary solution is valid. This question is solved by the CYCLE800 logic where the machine definition and kinematics including axis limits, _DIR parameter etc. are considered.

SiemensTCOFR

Function — ROTATION_PLANE

Status — Active

Comment — Added V7.3.4

Valid Inputs — Text, Value

This macro is used to calculate the rotary axis angles for the Working Plane (TCOFR command used in CYCLE800 sub). When the Override Value=1 and the Override Text value is a string specifying the names of variables where the results are stored, the macro calculates both sets of angles and transfers the result to specified variables, including the number of solutions. The Override Text value for use in CYCLE800 should be “\$P_TCANG[1] \$P_TCANG[2] \$P_TCANG[3] \$P_TCANG[3] \$P_TCSOL” where the first 2 values (index 1 and 2) are used for the first solution of Carrier and Rider angles, the next pair (index 3 and 4) is used for the second solution or Carrier and Rider angles, and the \$P_TCSOL is used to store the number of

solutions (usually 2 when 2 rotary axis are present). The calculated values are theoretical, not restricted by machine limits or other requirements.

This function was performed by the **SiemensPAROT** macro in its previous version. When the macro is called with Override Value=0, all specified variables are reset to 0. This is usually done when the variable TCARR is set to 0. If the CYCLE800 logic concludes that the applied solution is the second pair (\$P_TCANG[3], \$P_TCANG[4]) it will call the TCOFR command a second time with the "_SOL2" variable set to 1. In this case the macro will swap solutions replacing the values in 1,2 by 3,4 and vice versa.

SiemensPAROT

Function — ROTATION_PLANE

Status — Active

Comment — Added V7.3.4

Valid Inputs — Text, Value

0 = reset to the last applied working plane

1 = apply the calculated angles

This macro is typically called with the Siemens 840D PAROT or PAROTOF command.

When the **SiemensPAROTVersion** macro is called with Override Value=0 (default), this macro converts the "working plane" into real A, B, C angles that apply to the current machine, and establishes a rotation plane matrix if needed. The resulting angles are then stored in the specified variables. The text value passed to this macro contains the names of the variables, and the mode which is basically read from the Siemens 840D CYCLE800 parameter 4 (_MODE), for example "REALA REALB REALC #_MODE". This macro uses \$P_WPFRAME frame orientation and calculates the machine rotary component position using the WorkingPlane2AbcType macro. The calculated angles are stored in the variables assigned by the supplied names and can be used to position the machine so the tool, relative to the work piece, is oriented as specified by frame. An Override Value of 1 specifies that the calculated angles are applied, an Override Value of 0 is used to reset to the last applied working plane.

When the **SiemensPAROTVersion** macro is called with OV=1, this macro activates the new logic which calculates \$P_PARTFRAME frame based on rotary axis angles specified by variables in OT string parameter. When the PAROT command is called in CYCLE800 subroutine the selected solution is in local variables _A1, _A2 or system variables \$TC_CARR13[1], \$TC_CARR14[1]. In most cases OT should be set to "\$TC_CARR13[1] \$TC_CARR14[1]". Names REALA REALB REALC should not be used with the new PAROT logic. Note that block N820 in

TOOL_CARR subroutine (or CUST_800) should not be edited but used as it is in the machine arc file. Since solution selection now is performed in CYCLE800 subroutine there is no need to configure PAROT command with macros specific to solution choice, _DIR parameter and other factors. Macros **WorkingPlane2AbcVertRiderZero** and **WorkingPlane2AbcVertRiderOffset** should not be used anymore.

ToolOffsetActiveToolOnOff

Function — TOOL OFFSETS

Status — ACTIVE

Comment — Added V7.3.4

Valid Inputs — Value

0 = Off

Anything else = On

This macro turns on and off the special tool offset calculation associated with the active tool. The default is off.

The direction that a motion component moves is dependent on whether the component is on the Tool side, or the Part side. For example, a Z10 might move the tool up if the Z is on the tool side. Or it might move the part down if Z is on the part side.

On some complex machines (machines like the Citizen), sometimes the Tool is on the same side as the part. Now let's say the Z is on the Tool side, and the part is also on the Tool side. The Z is marked as a part side component, and is moved in the opposite direction as normal. Now if we want to apply a Z10 offset, we must apply a -10 offset since this component moves in the opposite direction as normal

This macro turns on a new special logic that automatically adjusts the tool offset based on the location of the corresponding motion component, the active tool component, whether the motion component is marked as part side, and whether the component is marked as "Reverse Direction".

Although this logic is valid for all machines, the default will be to continue with the current logic.

NOTE: The new logic requires the active tool to be valid at the time that the tool offset is applied.