



What's NEW in VERICUT 7.0

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

August 10, 2009

Dear VERICUT® User:

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

The VERICUT 7.0's NC program simulation, verification, and optimization technology is packed with new features making it more powerful and easier to use. This letter describes important changes in VERICUT 7.0. Take a moment to review what's new and improved in this release.

Maintenance and Licensing Information

How To Get a License - All users must complete and return the License Request Form in the CD booklet, or submit the application at <http://www.cgtech.com/usa/support/license/>. Licensing is sent via Email only.

NOTE: This software requires a VERICUT 7.0 license.

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

Sincerely,

Bill Hasenjaeger

CGTech Product Marketing

VERICUT 7.0

Release Notes

August 10, 2009

VERICUT 7.0 Enhancements

Highlights

The Project Tree is enhanced to reduce the number of pop-up dialogs that the day-to-day VERICUT user must navigate to set up a simulation. Setting up a simulation in VERICUT 7.0 is very different than VERICUT 6.2.

- The Project Tree must be used to configure a project.
- The features formerly in pop-up dialogs that are most commonly used for the day-to-day use of VERICUT are located in the Project Tree.
- An optional Configure menu appears at the bottom of the Project Tree for the selected branch in the Project Tree.
- A new file selection method is built into the Project Tree.
- Actions in the Project Tree Configure menus are applied immediately. There is no OK, Apply, or Cancel button to press.
- Machine components are optionally displayed in the Project Tree, thereby eliminating the Component Tree.

The basic philosophy of the Project Tree enables you to configure a project's setups using the Project Tree features as a guide, as follows:

- Use the Project Tree to step down through the tree structure, configuring each branch of a setup as you go.
- A "Configure" panel is optionally displayed at the bottom of the tree's dialog during setup configuration.
- The "Configure" panel displays settings and actions most commonly used for configuring the setup.
- The features on each "Configure" panel are specific to the branch or item selected in the tree.
- Filling-in text and numeric fields, checking options and pressing buttons in the panel cause an *immediate* action when used. There is no **OK**, **Apply**, etc.
- Less commonly used features are available in the right-mouse button shortcut menus associated with each branch or item selected in the tree or from pop-up dialogs accessed from the VERICUT Menu bar.

VERICUT uses a new method to calculate and animate the motion path from the NC data.

- Animated motion, in all view types, are coordinated.
- Tool images are consistent between views.
- Slow motion and skip cuts are identical for all motions and view types.
- Collision tolerance and motion display are independent of each other.

Machine and Control files are in XML format.

Enhancement Details

Verification

VERICUT is enhanced to support automatic corner rounding and chamfering lines and circles for all cases; approaching/departing, inside/outside, and CW/CCW.

The Toolbar is now fully customizable. You can now add or remove individual icons and define the order in which they are displayed in the toolbar.

Material removal, and Tool display, in the Workpiece view are enhanced to support adding an offset Driven Point (in X or Y) to a Milling Cutter.

Water jet tool display is enhanced in the VERICUT graphics window to display the three different ranges of the tool differently, similar to the way that water jet tools are displayed in the Tool Manager Tool Display area.

Block parsing/processing is enhanced to enable applying a new variable value on the same block in which it was calculated.

The "Preserve Stock Transition" button is moved to the Project Tree's cut stock model's right-mouse button menu and to the cut stock's Configure Model: Assemble tab.

VERICUT is enhanced to enable referencing an NX part file. VERICUT will open and import the part file via a NX/Open application.

VERICUT Machine and Control files syntax is now in XML format.

The Working Directory is now saved in the user preferences file.

The ability to rename a coordinate system in the Project Tree is added to the coordinate system right mouse button menu.

The Project Tree is enhanced to enable specifying G-Code Offsets (Program Zero, Work Offsets, etc.) within the Project Tree.

New feature, “Auto-set working directory to the current project folder”, is added to the Preferences window to enable automatically setting the Working Directory to the project file’s location.

New feature, “Suppress save and display thumbnail image”, is added to the Preferences window to enable suppressing the creation of the thumbnail image during the saving of a Project file and suppress the display of the thumbnail in file selection windows (Open, Save As, etc.).

A model highlighted in the Project Tree can now be un-selected by clicking in a blank spot (background) in the VERICUT graphics area.

Right clicking on any file “Save” icon (Save Project, Save Machine, etc.), changes the icon and behaviour to “Save As” mode. The state of the toggle is saved in the user preferences file.

The NC program file name is now written to the VERICUT Log File when an Error or Warning is reported.

Tap error messages are enhanced to enable more quickly identifying the problem (feed speed or tap definition).

"Error: Tap cycle advance 'xxx' is incorrect for tap tool 'yyy' pitch at line 'zzzz'".

Where: 'xxx' = feed distance per revolution.

'yyy' = tool ID.

'zzzz' = line number.

A right mouse button menu has been added to the Message Logger in NC Program Review mode to enable selecting whether Errors, Warnings, or both are displayed. When active (checked) Errors and Warnings are now both associated with the other NC Program Review windows as Errors were in the past.

Collision checking between Tool Holders the Stock/Fixture in a Workpiece View is no longer dependent on the holder being visible. Collision Checking settings and Tool/Holder visibility setting are now independent of each other. Tool/Holder to Stock/Fixture collision settings are now set in the Project Tree, Check Collisions Between menu accessed by clicking on a Collision branch. Cutter/Holder Visibility settings are set in the Project Tree, Configure Tooling menu or from a Tooling branch right mouse button menu.

Design component visibility in a particular view is now based on its based on its Visibility setting (Blank, Machine View, Workpiece View, or Both) except while cutting.

Syntax Check is enhanced to enable checking multiple NC programs at one time. In the Syntax Check window (Info > NC Program, Utilities menu > Syntax Check), The “Check Active” button checks the NC program that is currently “active”. The “Check All” button checks all NC programs, sub-programs, and subroutines referenced by the current setup.

A new function, atan2_d0to360, is added to returns values in the range of 0 to 360 degrees to match the capabilities of Fanuc controls.

atan2_d0to360

atan2_d(yvalue, xvalue)

This function returns the arc tangent of yvalue/xvalue. The returned value is in degrees, and is in the range of 0.0 to 360.0. If yvalue and xvalue are both zero, the return value will be zero.

A new function, asin_d270to90, is added to returns values in the range of 270 to 90 degrees to match the capabilities of Fanuc controls.

asin_d270to90

asin_d270to90(value)

This function returns the arcsine of the specified value. The specified value must be in the range of -1.0 to 1.0. The return value is in degrees, and is in the range of 270.0 to 90.0.

Dynamic Controls is enhanced to support PowerMill mouse action conventions.

Pan = Shift key + Middle Mouse Button

Zoom In = Mouse Wheel forwards (away from the operator)

Zoom Out = Mouse Wheel backwards (toward the operator)

Dynamic Rotate = Hold Mouse Wheel down

Project Tree is enhanced so that a unique name is assigned when a new setup is added.

The MDI window is enhanced so that once you activate the NC Block Entry text field by clicking in it; it remains active enabling you to continue adding blocks without re-activating it.

A new “Do Not Shorten Cutter” feature is added to Calculate Min. Cutter Extension that that when active, prevents VERICUT from changing the cutter height to be shorter than originally defined in Tool Manager.

The ability to optionally display the tool path line display in a Machine View when in NC Program Review is added.

The cutting motion in both the Workpiece View and the Machine view are now synchronized.

The Animation Speed slider is enhanced to enable controlling animation speed as well as setting Block Skip values.

The Project Tree is enhanced enabling you to unselect models in the Project Tree by clicking on a blank space in the VERICUT graphics window.

The Design component, by default, is only visible in a Workpiece view. It can be made visible in other views.

VERICUT is enhanced to enable assigning a variable and using the variable in the same block.

Design components that are automatically moved to a subsequent setup are now restored to the original setup when the project file is saved.

Fixture collision errors are enhanced to include the fixture component name.

A new Drill Cycle processing option, "No Motion" is added to enable drill cycle processing and material removal without animation.

Machine Simulation

An Ignore column is added to the Machine Settings: Collision Detect tab, similar to the one on the Machine Settings: Travel Limit tab, enabling to toggle checking "on" or "off" for each collision record. The on/off condition of the toggle is only retained during the current VERICUT session.

The Tool Change Retraction table is enhanced to enable controlling the order in which axes move to the tool change location.

Collision Detection is enhanced to highlight the component collision pairs, including sub-components when checked, when a row is selected in the collision table. The components are highlighted by shading them in the "error color". The highlight is removed when the tab is not raised and when the dialog is closed.

The Machine View, right-mouse button, Component Visibility list no longer contains components that do not have models attached.

Machine selection has been enhanced by the addition of the following features:

- When a machine file is saved, a thumbnail image of the contents of the first Machine View is also saved.
- The machine thumbnail image is displayed in the file selection window while browsing to open a Machine file, in the same manner that a thumbnail image is displayed in the file selection window when browsing to open a Project file.
- The machine thumbnail image is displayed when the cursor is held over a machine file in the in the Machine File pull-down list in the Project Tree, Configure CNC menu.

Machine Simulation is enhanced to behave more like the machine for situations where the tool assembly does not include a cutter (spin the holder, continue logging errors, etc.).

Collision checking is enhanced to enable setting a "near miss" value for components and the cut stock.

Collision reporting is enhanced so that if Animation Speed is slow, and Stop At Max Errors is toggled "On", simulation will stop at the intermediate point where the collision occurs.

Tool Manager

The ability to “stack” tool assemblies by default is returned to Tool Manager for all non-turn tool types.

The tool display in the Tool Manager is enhanced to shade the cutting portion of the tool differently than the shank portion of the tool.

The Turret Aid window is enhanced so that all settings are retained during the current session.

The file selection field is enhanced so that if a file name is specified without a path, Tool Manager searches for the file in the same three default paths that VERICUT uses (Working Directory, parent of Working Directory, Library).

Tool Manager is enhanced to display both “active” and “inactive” OptiPath records associated with the tool.

A “rapid” search tool is added to the Tool Manager Menu bar.

X-Caliper

X-Caliper Distance/Angle output is enhanced to enable measuring the polar angle of a cylinder axis from a defined center point and orientation.

X-Caliper is enhanced to retain the display and orientation of “From”, “To” and measurement markers during dynamic view changes (rotate, zoom in/out, pan).

OptiPath

Depth limit checking is enhanced and a warning message is output if the cutting depth exceeds a specific amount.

G-Code Processing

New macro, GlobalVariable is added to enable sharing variables, or variable ranges, across subsystems.

Right clicking on the Step VCR button displays a new feature enabling you to specify how VERICUT is to handle control subroutines encountered during processing. Choices include Step, Step Into Subroutine, Step Over Subroutine, Step to End of Subroutine.

Macro DynamicToolTipOnOff is enhanced to handle any turning tool, including flash tools with single insert, in the turning plane adding support for dynamic control point during multi-axis turning.

Support is added for true helical material removal when processing G-Codes, including a new X-Caliper feature "helix sweep".

Two new macros, SetCycleFeedrate and SetCycleFeedrate2cycle, are added to support a main and secondary feedrate that is specific to drill cycles.

The Axis Priority logic is enhanced to allow for multiple groups. Each axis can now have an unlimited number of priorities designated by the group number. By default, each axis has a priority group designated as "0" and may not be deleted. All other groups can be added or deleted. The priority group can be change at anytime during the simulation via the AxisPriorityGroup macro.

Support is added for multiple line comments. Two Special words: Multiline Comment, defining the comment start (for example, /*) and End Comment, defining the end of the comment (for example, */) must be defined. It is also important to remove the "/" word used as a skip character, and all workaround words, settings, macros etc.

A new capability of adding comments to describe what Word/Value pairs do in the control is added. These comments are also written out to G-Code and Control Report files.

New macros XRelationalOffsetCompName, YRelationalOffsetCompName, ZRelationalOffsetCompName, and RelationalOffsetRegisterName are added to support Relative From/To offsets for axes other than XYZ axes.

Support is added for the Siemens 840D variable type AXIS.

DEF AXIS

DEF AXIS will create a word of type "Special", with subtype "Variable Name". This word will be marked as being created in the current subroutine (or main), and be removed when exiting from this subroutine.

DEF AXIS will also create a variable of type "AXIS" or "axis array". This variable will be marked as being created in the current subroutine (or main), and be removed when exiting from this subroutine. All variables will be treated as global.

The following "AXIS" formats are supported:

```
DEF AXIS name
DEF AXIS name_1, name_2, name_3
DEF AXIS name_4 = X
DEF AXIS name_5, name_6=C, name_7=SET(Z)
```

The following "axis array" formats are supported:

```
DEF AXIS Name[m]
DEF AXIS Name[m] = SET (X, Y, Z,...)
Name[i] = SET(name1, name2[j],...)
Name[i] = REP(name1)
```

The actual axis variable is a string representing the AXIS name and its size is used to allocate the necessary space. Internally the AXIS variable is a string limited to contain only supported axis names.

- Commands SET and REP can be used with both simple axis variable or axis array.
- Parameters for SET, or REP commands can be an axis (constant), a simple variable, or axis array element. No axis expression is allowed.
- Word SET should be added to Word Format table (Type = Special, Sub Type = Sin840D SET).
- Word REP should be added to Word Format table (Type = Special, Sub Type = Sin840D REP).

Rules For Using:

1. DEF must be followed by one or more spaces or tabs.
2. AXIS must be followed by one or more spaces or tabs.
3. The end of the variable name is marked by a space, tab, comma, semi colon.
4. The end of a variable definition is marked by a comma or semi colon.
5. One or more spaces may exist prior to the =.
6. One or more spaces may exist after the =.
7. The left parenthesis will indicate an initial value. The right parenthesis will mark the end of the initial values. Expressions will not be allowed. The constant value must be a valid axis name without any quotes (exactly as it is used with motion commands (X5 Y10)).
8. Supported axis names are all 840D axis (X, Y, Z, A, B, C, U, V, W, A2, B2, C2, X1, Y1, Z1, X2, Y2, Z2, S, S1, S2 and S3). Any axis used as axis variable must be specified in Word/Format table as "Macro" or "Conditional".
9. Since VERICUT supports only 12 axes, the first 12 axes can refer to corresponding axes in VVERICUT. However the NC program can reference any Siemens 840D axis if it is specified in Word/Format table.
10. '[' and '(' must be defined as Left Precedence.
11. ']' and ')' must be defined as Right Precedence.
12. Everything beyond the semi-colon will be ignored.
13. For more details see: Job planning 1-4 Programming Manual, 03/2006 Edition or later.

The concept of axis variable is:

1. Axis name can be stored and referenced without being a string.
2. Numerical array elements can be referenced by axis name instead of index (where X is equivalent to index = 0).
3. Axis variable or constant can be used in logical expression and in function argument list.

VERICUT is enhanced to support these features. Following are some examples.

; Axis definitions:

```
DEF AXIS _AAX = Y ;Y axis assigned to variable _AAX  
DEF AXIS _AAY = X ;X axis assigned to variable _AAY
```

```
DEF AXIS _AAZ = Z
DEF AXIS _ALLAX[12] = SET(X,Y,Z,A) ;_ALLAX[0-3] initialized to
X,Y,Z,A
```

; supporting definitions

```
DEF REAL ABC
DEF STRING $P_AXNAMs[12] ; string array
DEF REAL $P_AVAL[12] ; REAL array

_AAX = _AAY ; X assigned to _AAX
_AAX = X ; X constant assigned to _AAX (no change)
ABC = 5.5
$P_AXNAMs[X]="X" ; string "X" assigned to string array element
$P_AXNAMs[0]
$P_AVAL[Y] = 12.5 ; value 12.5 assigned to array element
$P_AVAL[Y] (or $P_AVAL[1])
_ALLAX[5] = _ALLAX[2] ; Z is assigned to axis array element
_ALLAX[5]
_AAY = _ALLAX[1] ; Y is assigned to _AAY
$P_AVAL[_ALLAX[2]] = 22.5 ; value 22.5 is assigned to $P_AVAL[Z]
(or $P_AVAL[2])
$P_AVAL[_AAX] = 2.5 * $P_AVAL[Y] + $P_AVAL[_ALLAX[2]] ;
$P_AVAL[X] value is 53.75
ABC = 2.5 * $P_AVAL[Y] + $P_AVAL[_ALLAX[2]] ; ABC value is 53.75

IF (ABC == 5.0); condition not satisfied
ABC = 3.3
ENDIF

IF (X == _AAX AND Y == _ALLAX[1]); condition satisfied
ABC = 2.2 ; ABC value is 2.2
ENDIF
```

```
GEOAX (ABC, _AAX); function call with axis argument as a variable name
GEOAX (ABC, Y); function call with axis argument as a constant
```

; axis variables and constants in frames

```
$P_UIFR[2] = CTRANS(_AAX,22.5, _ALLAX[1],22.5, _ALLAX[2],145)
$P_UIFR[3] = CTRANS(X,45+22.5*COS(30), Y,22.5, Z,145+22.5*SIN(-
30)):CROT(_AAZ,45):CROT(_ALLAX[1],30)
```

A new conditional function, SiemensAXISCond is added to support AXIS constants in logical expressions when an IF statement is processed. All axes used in the NC program and referred also in logical expressions should be defined in Word/Format table using SiemensAXISCond conditional function and format specifically for particular axis.

Support is added for Siemens 840D AC command for I/J/K on a G2/G3 command while mirroring.

Support is added for processing Okuma turning cycles with Cutter Compensation.

The SyncValue macro is enhanced to apply to all input channels. If a text value of "ALL" is passed, the current input channel will sync with all other input channels.

New macros MSWriteToFile, MSCloseFile, and MSOpenFile are added to provide a more generic application of the existing FanucDprint, FanucPclose, FanucPopen macros and work in exactly the same way.

New macros AxisMotion and AxisMachineMotion are added to enable redirecting to the appropriate macro implementation (XAxisMotion for AxisMotion with OT="X", AAxisMachineMotion for AxisMachineMotion with OT="A", etc) with the override value passed along. Valid Override Text values are "A", "B", "C", "A2", "B2", "C2", "U", "V", "W", "X", "Y", "Z". Macro AxisMotion should not be used when you need to specify the override text "-" to select the motion direction in absolute mode.

A new override value, 99, is added to macro WorkingPlane2AbcType to provide a "Universal" type similar to the "Universal" type previously implemented for the Ijk2AbcType macro.

A new macro, TravelLimitErrorReporting, is added to enable specifying whether or not travel limit errors are reported during rapid motion.

A new macro, CirclArcAngle, is added to support G2 and G3 with a pitch and angle in Heidenhain MillPlus helical motion.

Three new macros, SetRelWorkCoordFrom, SetRelWorkCoordTo, and SetRelWorkCoord are added to support advance features like RPCP, that require that the work offset table to be relative (ex: Tool to Stock) when the work offset is created from within the NC program.

The G-Code 5-axis motion break-up, created for material removal, is enhanced so that intermediate points have the correct (matrix) orientation in space.

Words A40=, B40= and C40= are added to the HeimPlus Library control and call the IgnoreMacro macro, to prevent them from being interpret as A rotate 40 degrees, etc.

Support for Cycle 204 is added to the Heid530 Library control.

Word CCA is added to the Heid530 Library control and calls the TangentialAngle macro.

G84.1 rigid tapping support is added to the Acramatic 2100 Library control (acr2100.ctl).

Support is added for Polar Interpolation, for situations where C is not at zero, to the Fanuc 15t Library control (fan15t.ctl).

Support is added for Heidenhain Mill Plus G02/G03 helical motion with a pitch.

Miscellaneous

Batch Wizard is enhanced to enable specifying a Working Directory where output files are to be written to.

Batch Wizard is enhanced to enable re-sizing the Main window and the Editor window to be able to see the entire command line.

NC Program Review is enhanced to display the tool path trace in a Machine/Cut Stock View when Animate Machine When Stepping Back is toggled "on" in the Properties window.

All APT Sample files are moved to a separate directory.

Setup Plan is enhanced so that any visible coordinate systems are shown in the Setup Plan report picture capture.

The right mouse button menu for models is enhanced so that Visible and Delete are no longer next to each other.

The tilde symbol “~” is now recognized by VERICUT as a valid APT character.

An option is added to the VERICUT installer to VERICUT Limited shortcuts to be created.

The Display HOSTID utility is enhanced to show up to ten devices and now uses Sentinel LM 8.2.1.

Support is added for 32 bit and 64 bit Vista.

The .bat files in the /commands/ directory are enhanced to enable the recognition of variables defined prior to calling the .bat file.

The tool display in Tool Manager and in Machine views are now updated when the Model Tolerance value changes.

Many sample files are enhanced and new sample files are added.

VERICUT Help has been restructured to focus on the need to use The Project Tree for setting up VERICUT projects.

CATIA V5-to-VERICUT Interface (CATV5)

The CATV5 "Options" dialog is enhanced to enable setting different chordal deviation values for design, stock and fixture models. This is not a tolerance per model, but a value per component type, and applies to all setups.

CATV5 is enhanced to enable selecting STL files used in the CATProcess file as STOCK, FIXTURE, or DESIGN and passing the file reference and position through to VERICUT.

A "Merge to tool library referenced by setup template" option is added to CATV5 to enable merging tools from the current setup with those in the tool library referenced by the Setup Template.

CATV5 is enhanced to enable the Design component to follow the Cut Stock from one setup to the next. If the Design component is selected for the first "active" setup, it is attached to the Stock component and will follow the Cut Stock from one setup to the next. If the Design component is selected in any other setup, it is attached to the Fixture component and will NOT be passed from one setup to the next.

A check box labeled "Generate one tool library for all setups" is added to CATV5. When toggled "on" (checked), all tools with unique IDs from all setups (active or not) are placed in one library, with its name the same as the new project (but with extension ".tls").

The CATV5 "Options" dialog is enhanced to enable specifying a subsystem for the Work Offset table.

CATV5 is enhanced to keep track of the last "Generate tool library from CATIA information"/"Use specified TDM Tool list" setting as the default setting when accessing a new CATProcess file for the first time.

CATV5 is enhanced to enable passing Tool Holder IDs from the Process Resource List through to VERICUT.

CATV5 is enhanced to enable merging the tools from the current CATProcess file into the tool library referenced by setup template.

MasterCAM-to-VERICUT Interface (MCAMV)

Support is added for MCAMV on Windows XP64.

A French mcRes.local file is now provided with the MCAMV installation.

MCAMV is enhanced to work with MasterCAM X4.

NX-to-VERICUT Interface (NXV)

(formerly known as the Unigraphics-to-VERICUT Interface (UGV))

NXV is enhanced to enable specifying sub-programs.

NXV is enhanced to enable specifying the job subroutines that are to be in the VERICUT session.

NXV is enhanced to support new NX3 tool definitions.

NX4.bat, NX5.bat, and NX6.bat files are added to the commands directories.

Problems resolved in V 7.0

Verification

A problem causing a strange display of a particular facing head in both the Machine and Workpiece views is fixed.

A collision between the Z-axis model and the Cut Stock in a specific VERICUT project file is now correctly reported.

Invalid Tool/Stock collision errors and bad material removal are corrected for a two specific tools with unusual cutter shapes, used for unusual 5-axis motion.

NC Program Review now updates the tool location correctly during a circular motion in a specific VERICUT project file.

Unexpected VERICUT termination no longer occurs during material removal for a specific VERICUT project file using a VERICUT Solid model that was created with a very small tolerance.

A false holder collision is no longer reported for a specific tool holder profile that drops down inside of the cutter profile.

A collision between the holder and the Cut Stock during a drilling cycle in a specific VERICUT project file, displaying only a Workpiece view is now correctly reported.

VERICUT now processes the same number of loops that the machine does for a specific VERICUT user file.

Collisions between the holder and the cut stock are now correctly reported for tool assemblies that have the tool shank inside of concave bottom cutter.

Unexpected VERICUT termination no longer occurs for situations where VERICUT processing runs out of memory due to an invalid cutter profile.

It is now possible to construct a coordinate system on a Cut Stock using the Construct > Circle method. In some cases the pick will fail to resolve a CSYS and you will be prompted to try picking again in a different location.

Control file changes are now saved correctly from inside a specific multi-setup project file.

A lathe threading tool used in a specific VERICUT project file no longer flip-flops around the Z-axis, when the Animation Speed Slider is used to slow down the simulation.

The maximum character limit for IP files in the File Summary > Copy Files window is increased to enable using longer directory paths.

Volume is now correctly reported in the tooling section of a VERICUT Report when FastMill is being used.

Control Settings are now correctly displayed when working with IP files.

Project file models are now included in the File Summary when an encrypted machine file is being used.

Stop At: End of each Setup now works correctly when the X-Caliper window is open.

Adding a model to a translucent component now displays the model as translucent.

Setting up a VERICUT Report to display the Tool Diameter in the Tool Summary Table now correctly retrieves and displays the tool's diameter value instead of the tool's radius value.

The Field of View Angle slider on the View Attributes window: OpenGL Settings tab, Display Options tab is now correctly activated when Perspective View is toggled on.

Unexpected VERICUT termination no longer occurs when exporting a cut stock as STL file with the Reduce Triangle feature toggled on.

Driven Point values and display are now correct for round inserts.

Having the X-Caliper window open no longer deactivate the Project Tree right mouse button shortcut menus.

Collisions between the holder and the cut stock are now correctly reported when the Animation Speed slider is at 100% for a specific tool assembly that has a tool with an unusual shank shape.

The Cut Stock is now displayed correctly in both the Machine/Cut Stock and Workpiece view regardless of whether the spindle is spinning or not for a specific VERICUT project file.

The jaws no longer spin out of round, on a lathe using programmable jaws, once the spindle is turned on and the jaws are moved.

False Holder/Cut Stock collision errors are no longer reported when the Animation Speed slider is set at 100% for a specific VERICUT project file.

Material removal is now correct for a specific Parametric Cutter used in a specific VERICUT project file.

An error message is now output when trying to use a circular probe motion. Only linear probe motions are supported.

Mirroring a Sweep solid model relative to a coordinate system, now displays correctly when the project file is saved and then re-opened.

The X-Caliper Air Distance value is now correct when measuring after a specific roughing operation.

Material is no longer left behind for a specific tool used for 5-axis cuts around the inner and outer periphery of a pocket.

Unexpected VERICUT termination no longer occurs when outputting a VERICUT report file in HTML format using Japanese characters.

Shank profile removing material errors are now correctly output for a positive Z motion using a specific tool having two shanks. One is the non-cutting portion of the cutter above the flute length. The other is a small disk placed in the concave bottom portion of the cutter.

Warning messages, including the Line Number that triggered the warning, is now displayed in NC Program Review message area to enable clicking on a warning message and have VERICUT jump to the location of the warning in the NC program listing and in the graphics area in the same way that clicking on an Error message does.

The collision is now correctly reported when the Collision Detect table record is set to check for collisions between X component and the Z Table component (with sub components) and the Near Miss value is set to zero.

The VERICUT session no longer “hangs” for a specific VERICUT project file when Scan NC Program Files is toggled on in the G-Code Settings window (now the Project Tree, Configure Setup menu: G-Code tab).

Unexpected VERICUT termination no longer occurs after loading a specific IP file and pressing Play without first pressing Rewind NC Program.

A 64 bit VERICUT session no longer hangs when processing a specific VERICUT project file.

Machine Simulation

The machine model visibility, for a specific machine, is now correct when the VERICUT project file is opened a second time.

A collision between a holder and the Stock, with the Stock spinning, for a specific VERICUT project file is now detected correctly.

Using the DwellSeconds macro in a sync job no longer causes a bad move for the head opposite the one that invoked the DwellSeconds macro.

The Driven Point is now correctly positioned on the selected insert of a multi-insert lathe tool regardless of the insert’s position in the tree.

The modified tool list displayed in Tool Manager now reverts back to the original condition when a Save Tool File As action is cancelled.

Milling Tool Wizard no longer permits manually changing the Gage Point when Automatic Gage Offset Z is active. An error message is now output under this condition. Turret Setup now displays tools in the correct orientation after editing a previously setup turret.

Tool Manager

Support is added for alpha-numerical tools on tool chain when only a cross reference tool list is used.

Holders created as parametric cones now display correctly in the Workpiece view.

X-Caliper

Feature/History records are now correct for helical holes.

The Volume values returned are now correct when using multiple setups.

AUTO-DIFF

AUTO-DIFF results no longer vary for situations where AUTO-DIFF was performed in a prior setup.

During constant gouge check, the color-marked design model indicating gouges/excesses in one setup is now positioned correctly in the next setup.

G-Code Processing

Axis array values can now be successfully used to define a Siemens 840D DEF real array.

New macros RestorePrevMotionType, RestoreSavedMotionType, and SetSavedMotionType are added to support the need to process a G28 command in rapid mode.

A new conditional, TosnucCondVWord is added to enable sharing an axis address as a variable tag.

VERICUT no longer slows, or hangs up, while processing large (large number of records) 2-axis lathe G-Code NC programs when “Replace Material When Stepping Back” is toggled on.

VERICUT now produces the correct circular motion and material removal for situations when in polar interpolation mode, with cutter comp set "on-default to zero" and C rotary changes direction.

The Tool Tip Y value in the Status window is now correct for situations where “Allow Motion Beyond Limit” is toggled off.

Variable round off differences no longer cause incorrect “IF” check results.

Siemens frame functions, CROT and CTRANS now work correctly after an IP file is “merged”.

Material removal when processing a G03 XW circle with ActivateAxis is now correct.

The NewCycleLogic macro is enhanced so that it is now set globally, eliminating the need for it to be called at start of processing for every subsystem.

The “universal” conversion type 99, for the Ijk2AbcType, and the WorkingPlane2AbcType macros are enhanced to use the spindle axis if no tool is loaded.

The “universal” conversion type 99, for the WorkingPlane2AbcType macro is enhanced to ignore the "spindle orient" rotary axis when determining the angles to calculate.

NumSubroutineSequence and similar macros now work with blank text entries.

Rotary Travel Limit Error values are now correct the first time that they are output.

A new Override Text value, “Order”, is added to macros NumSequence and SequenceStartEnd to support a NUM controls ability to create a loop where the Start Sequence block number is higher than the End Sequence block number. The macro NumSequence would pass this ORT value to macro SequenceStartEnd. If macro SequenceStartEnd is entered with ORT = "Order" the sequence start block number and sequence end block number are entered exactly in the order specified in the NC program block.

Miscellaneous

Installation of the Import/Export modules for STEP no longer requires the installation of the optional CATIA support.

The Tool Use graph now correctly displays the program record number.

The Output File Name can now be typed in, as well as using the Browse function, when creating PolyFix Converter batch script with Batch Wizard.

VERICUT is modified to add the directories containing the project and setup template files to the list of directories searched for the machine and control files. This should resolve the problem for NXV and for the other CAM interfaces when VERICUT is

started from one of the CAM interfaces and the machine and control files passed to VERICUT do not have directory paths.

VERICUT now consistently uses a yellow background when a menu option field requiring a mouse pick in the VERICUT graphics area is activated.

Coordinate System markers and Axes display are now consistent whether you are in OpenGL mode or in non-OpenGL mode.

Long delays are no longer experienced when loading a specific VERICUT Solid file (.vct) with 64 bit VERICUT.

VERICUT now prompts you to save the project file when only color settings have been changed.

Collision reporting is now consistent regardless of the Animation Slider position for tools that use a single point to define a concave bottom.

Using the new “Pause” feature, VERICUT can now be stopped in the middle of a large rotary motion without killing the process.

Using special characters in Control Notes no longer causes errors.

Creating a report while the Project Tree is open no longer interferes with graphical model selection.

Motion display is now consistent between the Machine view and the Workpiece view when simulating multi-channel machines.

Invalid errors are no longer output for tapping cycles when “Bottom Only” is active.

NC programs can now be activated/inactivated from within the project tree.

CATIA V5-to-VERICUT Interface (CATV5)

Prior CATV5 settings are now correctly retained when CATV5 is opened to start work on a new CATProcess file.

GibbsCam-to-VERICUT Interface (GIBBSV)

The \windows64\ folder now contains the correct Gibbsv.dll.

MasterCAM-to-VERICUT Interface (MCAMV)

MCAMV now works correctly with MasterCAM X4.

NX-to-VERICUT Interface (NXV)

Unexpected NXV termination no longer occurs when it is used with NX4 and 32 bit VERICUT on a 32 bit computer.

NXV, when NX4 is operating in a Team Center environment, now correctly fills the Table List, the 'From' List and the 'To' List pull down lists in the Options window.

NXV no longer outputs a series of error messages when Output Files is selected when the NXV window fields are only partially filled in.

Valid inserts are now created when NX User Defined tools (turning) are passed through NXV to VERICUT.

NXV now successfully passes Work Offset From/To settings to VERICUT.

The NXV "Merge Tools Into Setup Template Tool Library" feature now works correctly.

NXV is enhanced to handle a specific special NX tool definition.

New Macros in V7.0

The following new Macros are added for V7.0.

AxisMachineMotion
AxisMotion
AxisPriorityGroup
CheckVolumeAboveDepthLimit
CircleArcAngle
CollisionPreCheckOnOff
CompositeValue
ConvertArrayIndex
CycleTurnThreadStartAngle
DebugComponentAxis
GlobalVariables
MatchParensInComment
McallSubroutineOn
MSCloseFile
MSOpenFile
MSWriteToFile
RelationalOffsetRegisterName
RestorePrevMotionType
RestoreSavedMotionType
RollerFormCollisionIgnore
RotationPlaneXMirror
RotationPlaneYMirror
RotationPlaneZMirror
SetCycleFeedRate
SetCycleFeedrate2
SetSavedMotionType
TravelLimitErrorReporting
ViperCourseDist
ViperTowCut
ViperTowPinch
ViperTowPrefeed
ViperTowPrefeedDist
XRelationalOffsetCompName
YRelationalOffsetCompName
ZRelationalOffsetCompName

New Conditionals in V7.0

The following new Conditionals are added for V7.0.

TosnucCondVWord

New Functions in V7.0

The following new Functions are added for V7.0.

ISFILE

NOT

Macros not yet included in the documentation

See the [Macros not yet included in the documentation](#) section at the end of this document.

VERICUT 7.0.1 Interim Release

Release Notes

October 27, 2009

VERICUT Version 7.0.1 is available for all supported Windows and UNIX platforms.

V 7.0.1 contains everything described above for V7.0, plus the following additional fixes/enhancements.

Verification

Unexpected VERICUT termination no longer occurs for a specific two channel project file.

Unexpected VERICUT termination no longer occurs for a specific project file where the machine's initial position is outside of the specified Travel Limit values.

Cutter Compensation is now processed correctly when cutting two adjacent arcs during an Okuma rough turning cycle.

The Position and Angle values, on the Configure Component menu, are now consistently updated on all tabs (Translate, Rotate, Assemble, Matrix, and Csys).

Sweep and Revolve models can now be updated and saved in the Sketcher window without having to close and re-open the project file.

Unexpected VERICUT termination no longer occurs while zooming with a Space Pilot on a 64 bit computer.

Unexpected VERICUT termination no longer occurs when pressing Play after changing the Stop At setting, when the Stock is a VERICUT solid model.

Using the Orientation field in Tool Manager to correctly position a 90 degree head for mounting on the spindle, now processes correctly in VERICUT for circular moves over 360 degrees.

A problem, related to sync, circles, and a moving stock, resulting in the coarse display of a circle record, is fixed.

A problem causing the two "sync'd" turrets to collide after a tool change for a specific project file is fixed.

An error message is now output when switching into Polar Interpolation mode with inappropriate coordinate values. Sample message: "Error: Y -10, should be zero for PolarInterpolation".

A new command line option, "MinExtFile=filename", is added to enable calculating min cutter extension and save adjusted tools in a tool library named "filename" in Batch mode. For example, using command line option "MinExtFile=new_tools[S].tls" with vericut.Vcproject will produce the following three files: new_toolsS1.tls, new_toolsS2.tls, new_toolsS3.tls for three setups.

An icon is added to the VERICUT toolbar to access the Project > G-Code > Variables window.

The simulation now starts correctly, after a Rewind, with the tool specified using Start At = Tool ID.

An error message is now output when the maximum cuts stored (1000) for adding material back, when using the "Replace Material When Stepping Back", is exceeded.

A new feature, "Unbalanced parentheses on a single line", is added to Word Format window: Syntax Check tab which checks if the '(' and ')' characters are balanced on the same line, regardless of word type.

The User-Defined Tag Values window is now correctly populated when displayed at the time that a VERICUT report is created.

User-Defined Tag values modified time that a VERICUT report is created are now correctly stored in a setup in the project file.

VERICUT is enhanced so that when going from a Single View Layout to a Two View Layout, the added view type is determined as follows. If the single view is Workpiece View, the added view will be a Machine/Cut Stock view. If the single view is a Machine View or a Machine/Cut Stock View, the added view will be a Workpiece View.

A new option, **Add a View**, is added to the Right Mouse Shortcut Menu that displays when clicking with the right mouse button in a view in the VERICUT graphics area.

The Volume Removed values that are displayed in a VERICUT Report are now consistent regardless of the "Stop At" setting used.

In the Check Syntax window, the color of the lines reporting a missing ')' after a 'Multi-line comment' is now displayed correctly when the starting line of multi-line comment is not visible.

The Cutting Time values displayed in the VERICUT Log File and in a VERICUT Report are now consistent.

When running an NX CLS file, with the Tool change method is set to be Tool Name, the initial VERICUT scan now recognizes this as a valid tool change method and correctly changes the cutting tolerance to be based on Tool Size.

STEP files are now loaded correctly if the file extension is either .STEP or .STP.

The options available in the File End pull-down list is now consistent between the AutoSave window: In Process , View Capture and VERICUT Solid tabs and the VERICUT Output Files window: In Process , View Capture and VERICUT Solid tabs.

The color of a coordinate system can now be successfully changed to black using the color pallet on the Configure Coordinate System menu: CSYS tab.

Tool inserts are now correctly oriented in the resulting image when using the Render feature in the View Capture window.

Double clicking on a .vcproject file that has been associated to vericut.bat in Windows, now correctly opens the project file with the Working Directory set in the project file.

Unexpected VERICUT termination no longer occurs for a specific project file when the Cutting Conditions graph is toggled "On".

The Workpiece view no longer "freezes" when doing a combined rotate/zoom with OpenGL turned on when CATIA Dynamic Controls are active.

VERICUT no longer "freezes" when using the Jog feature in the MDI window, when the Remove Material feature is toggled "On", and using a millimeter .vcproject file.

The Tab Removal feature on the Delete Detached Stock window now correctly removes the tabs when picking a cylinder.

The Info > NC Program window no longer closes when stepping into a sub-program when using an encrypted control file (.xctl). VERICUT will now keep main program displayed, and blindly step through the sub-program, until the return to the main program and then continue stepping in the main program displayed in the Info > NC Program window. The following informational message is output to notify the user what is happening while VERICUT stepping through the sub-routine: "Cannot display encrypted sub-program".

Using the Mirror feature defined with a CSYS is now correctly retrieved for the Fixture when opening a Project file.

The Undo button on the Assemble tab now displays correctly when Look and Feel is set to Windows, or Metal.

OK to Cut Into Fixture and Milling Maximum Depth settings are now correctly saved in the vcproject file.

When cutting, holders constructed from multiple profiles no longer have a portion of the holder displayed the color of the cutter (cut color).

A specific Cutter Grinder operation no longer processes slower than it did in V6.2.2.

A specific project file, displaying multiple Workpiece views, now updates all views correctly when processed with 32 bit VERICUT on a 32 bit computer.

Unexpected VERICUT termination no longer occurs after processing a specific project file several times and then clicking in a Workpiece view when processed with 32 bit VERICUT on a 32 bit computer.

Creating a block model using the Block from NC Program Extent now correctly creates a block model and updates the values in the window.

Using the CSYS from File feature, on the Configure Coordinate System menu, now updates the Project Tree immediately.

The Project filename is now updated in the Project Tree immediately after using File menu > Save As.

Unexpected VERICUT termination no longer occurs when processing a specific, very large, project file that uses AutoSave > In Process to output an IP file at the End of each File.

Material removal is now correct for a specific project file that uses a Profile Cutter that contains some very short arcs.

Cutter Compensation now works properly when called within an Okuma G85 Turning Cycle for a specific project file that has both a Tool Cutter Compensation value and a Tool Nose Compensation value being applied.

Unexpected VERICUT termination no longer occurs for a specific project file when the Cutting Conditions graph is displayed.

File open errors without details are no longer output when opening a project file that contains model filenames that are an empty string.

The comment record, PPRINT/VERICUT-MOTION OFF, now works correctly.

The "Tip" that displays when holding the mouse over a model file in the Project Tree no longer gets partially hidden behind an OpenGL view when the length of the tip extends beyond the Project Tree and into the graphics area.

VERICUT no longer allows using the right mouse button menu in the Project Tree to select multiple coordinate systems to be the "active" coordinate system.

When selecting multiple NC programs, the order in which the files are selected is now maintained when they are added to the Project Tree.

Setting Stop at Text using the menu that opens by right mouse click on the Play/Start At Options button no longer requires hitting "enter" to save the entered text string. The text string is saved when the menu closes.

The highlighted directory in a File Selection window is now display at the center of the directory tree display rather than at the very bottom.

Rotary intermediate points are now correctly recognized and processed when no models are attached to the rotary component.

Machine Simulation

Encrypted machine files (.xmch) now load correctly when the associated STL files are located in a different directory than the encrypted machine file.

A new Siemens 840D specific macro, Siemens840DGoto, is added to support unconditional jumps to all GOTO destinations (labels, sequence numbers, expressions, or ...) supported by the Sin840D control.

The CornerMode macro now supports Override Value = 3, where the CornerValue is interpreted as the length of the chamfer to support the Siemens CHF command. You need to define a CHF "Word", and have this word call the CornerMode macro with OV=3.

The Stock position is now correct after doing a Reset when using a specific vertical turning machine with a pallet changer.

The following Fanuc Sync MillTurn display/status issues are resolved by the new SYNC logic in V7.0.1:

- Machine view is not aligned with the Workpiece view.
- The machine jumps down to get aligned with Workpiece view.
- Graphics are not updated/removed correctly.
- Sub-spindle Y position is incorrect.

Blocks containing a P-word sync code are now processed correctly for a specific Okuma OSP control.

Processing an M3/M4 no longer causes a facing head components to spin off center due to the presence of a milling tool component under the facing head spindle.

Fixture collisions are now correctly reported when OK to Cut Into Fixture is active, and the Maximum Milling Depth value is exceeded, while cutting along the tool axis or the centerline of Fixture model.

"Ok to Cut Into Fixture" is enhanced to override settings in the Collision Table. The following cautions should be noted:

- If a Fixture component has "OK to Cut Into Fixture" active, no Machine Simulation collision checking between a cutter and this fixture will be done.
- If a Fixture component has "OK to Cut Into Fixture" active, but the fixture is only visible in a Machine view, the "Max Milling Depth" parameter is ignored. Even if the cutter/fixture collision is deeper than the "Max Milling Depth", no error will be reported.
- The "Max Milling Depth" parameter is only recognized by VERICUT if the fixture is visible in a Workpiece view.

"OK To Cut Into Fixture" is enhanced to not allow RAPID motions. This enables seeing an error whenever a cutter RAPIDs into a fixture, no matter the amount.

A specific project file using a complex Swiss MillTurn (TSUGAMI TMU1R 10ax Horiz. Mill-Turn / FANUC 16iTB) machine, that processed correctly and transferred the Cut Stock correctly to the sub/back spindle in V6.2.2, now processes correctly in V7.0.1.

A specific project file using a MAZAK eRAMTEC V8II 6ax Vert. Mill-Turn / MAZATROL Matrix control no longer reports false counterbore holder collisions.

The Driven Point Axis now displays correctly when using the PivotOffsetCompName macro.

Component names and Csys names containing trailing blank spaces, now process correctly when used in a Work Offset table.

Using Rewind, then Play, after a pickoff motion in a specific project file now works correctly.

Machine Simulation can now be toggled On/Off after VERICUT is stopped in the middle of processing.

Tools are now displayed correctly when returned to the tool chain for a specific project file.

The turning CutStock for a specific project file is now correctly displayed in the Main spindle when unclamped after the CutStock is cut in two pieces.

SYNC now processes correctly for a specific project file that uses an Okuma control.

Tool Manager

Unexpected VERICUT termination no longer occurs for a specific G3 code and cutter combination.

Support for displaying tools with alpha-numeric Tool IDs on a tool chain is added.

Support is added for using a local OptiPath record on a referenced tool. When a referenced tool is loaded, OptiPath records associated with the externally referenced tool will be copied and marked as external references. You can add a local OptiPath record to overwrite an external OptiPath record. In Tool Manager, if all of a tool's matching OptiPath records are from external reference tools, the OptiPath main node icon displays black and white (the same icon used by "inactive" OptiPath records) and its tip says "Referenced OptiPath Record". You can not open/modify/cut/copy/paste a referenced OptiPath record.

Default model tolerances for "stand alone" Tool Manager are changed to 0.001 for inch and 0.025 for millimeter to produce more accurate tool images.

"Shank" tool components created in V6.2 now display correctly in 7.0.1.

X-Caliper

A new feature "Circle Center" is added to the X-Caliper: Distance/Angle tab to enable measuring model features relative to round objects in fixtures, such as tooling pins/holes, etc. After selecting this feature, you are prompted first to pick the XY plane and then a cylinder/cone face, similar to the CSYS "Circle" feature.

X-Caliper now measures hole depth correctly regardless of the drilling method used.

Hole depth can now be successfully measured, using X-Caliper, for a specific project file when a particular CSYS is active.

OptiPath

OptiPath no longer outputs an F0. feedrate, or duplicates the last line, in the optimized file for a specific VERICUT project file.

OptiPath is enhanced to use the same */path/filename* specified for the optimized NC program file for optimized subroutine files.

A new macro, OptiEOLStr, is added to enable defining a character string that is to be output at the end of every line in an optimized NC program file.

OptiPath time calculations are now correct when using the "mazak_variaxis730_maz640m.sub" tool change subroutine file from the \samples\Mazak\ directory of your VERICUT installation.

The optimized feedrates for a tool used in a specific NC program file no longer show abnormal feedrate variations.

The Search OptiPath Record window is enhanced to enable adding an OptiPath record from another tool library to a tool in the current tool library. An external record needs to be copied to the current tool library first and then it can be assigned to a tool. The following summarizes the new functionality:

1. An "Include Current Library" checkbox is added to the Search OptiPath Record window. This feature is also added to the Search Tool window.
2. A "File" column is added to the OptiPath Search result table which shows which file the record is from.
3. In OptiPath Search window, when an external record is selected, the "Duplicate" and "Close" buttons become sensitized. The "Duplicate" button makes a copy of the selected record; change the record's Material and Machine to the current setup's Material and Machine; and then insert the copied/modified record to the current tool library in Tool Manager. If you have "Include Current Library" checked on, you will see an orphaned new record displayed and highlighted in the Search OptiPath Record window.
4. In Search OptiPath Record window, with a local tool record selected, the "Add/Modify", "Reference", "Delete" and "Close" buttons are sensitized. Following Step 3, with the newly copied record selected, you can click on the "Reference" button to add the OptiPath record to the selected tool in Tool Manager. The "Reference" button will be de-sensitized if the currently selected tool is not optimizable (like a drill tool, etc.).

In summary, in order to reference an external OptiPath record, you first need to be "Duplicate" and then "Reference" it.

The OptiPath Savings Calculator, implemented in V7.0, now correctly displays time as Hours:Minutes:Seconds.

The "Clean-up Feedrate" is now applied during optimization when material has already been removed for simple circular motions.

The Cutting Conditions graph now displays the optimized graph correctly after doing a "Reset".

OptiPath now processes nested subroutine calls.

AUTO-DIFF

Constant Gouge Check is now ignored for Tap tools, thereby eliminating the false errors that were previously output for Tap tools.

G-Code Processing

The location of the Stock model, for a specific project file, no longer changes after saving and then re-opening the project file.

The support of the Siemens 840D MCALL command is enhanced to work with CYCLE type subroutines where parameters are set by calls to PROC type subs (i.e. with parameters) and the parameters are kept for the next motion. The macro McallSubroutineOn, implemented in V7.0, is used in the control to support this feature. McallSubroutineOn, followed by Siemens840DProcCall, is called on SubroutineName with MCALL; MCALL by itself triggers CycleSubroutineOff (which now resets the MCALL data).

G2 turning motions on multi-turret machines are now correct.

G-Code processing is enhanced so that a group of three IJK addresses are considered each time an I, J, or K address is called. Any address not explicitly set is set to empty. For example if J2K3 is called, VERICUT considers this as I J2K3, where the "I" address is set to empty.

False collision errors are no longer reported for special cases where the break down of circles during processing, result in two intersecting coplanar triangles.

The Word/Address "Find" utility is enhanced to search for search for variables and conditional states. In addition, when searching in the Word/Range, or the Variable tabs, the search now finds all occurrences in the Word/Address table, not just group records. For example, when the Word is "G" and the Range is "4", the group record G4 and the condition (G4) are both found.

Support is added for using the Siemens Sin840D function, CMIRROR, with Frame variables.

The Sin840D Library control is enhanced to support using the MillThread macros with the NewCycleLogic.

The Sin840D Library control is enhanced to support the command MCALL Lxxx, where the L word is suppressed from Word/Format and Lxxxx is treated like a PROC.

Support is added for the Fanuc command SETVN. The word "SETVN" needs to be defined in the control file, with the new special type "Fan_Setvn". Also, a new Special Sub Type "Fan_Setvn" is added to the Word Format table to support this feature.

The Call Stack option no longer incorrectly changes the first line NC Program Record to be the same as the second line NC Program Record when a third level subroutine is called.

The "universal" Ijk2AbcType = 99 now works correctly when multiple tool components are defined on the machine.

When a drilling cycle is processed, the macro CycleRapidLevelValue now recognizes when ZAxisZeroTracking is in use and adds the W-axis value so that the drilling Part Surface is correct.

Support is added for using an implied GOTO in an IF statement for TOSNUC controls.

Working Planes are now correctly rotated when using the "universal" option 99 with macros Ijk2AbcType and WorkingPlane2AbcType.

The macro, LabelMacro, is enhanced to support NC program records that contain label names directly after (no space) the block number, like those that are valid for Siemens 840D controls. For example: N795ALI_1

Unexpected VERICUT termination no longer occurs when VERICUT is Reset after converting a Text variable to a Number variable.

A new macro, AdjustToolOffset, is added to enable a G43 to apply tool offsets associated with a head rotation to be applied to the tool tip.

ReferencePointIndex and ReferencePoint macro processing is updated to execute the reference point part of a move in absolute mode, and then restore the mode afterwards.

When adding a new Word/Range pair in the Word/Address window, the macro is now correctly added to the new Word/Range created for situations where a conditional exists.

Cycle time is now consistent regardless of whether "Apply Acceleration to Cycle Time" is toggled On, or Off.

The message output for NC Codes that are not configured in the control is changed from "is not supported" to "is not configured in this control".

Executing a Fanuc G112 Polar command no longer causes the false error message, "Error Y should be zero for PolarInterpolation" to be output.

Using Save-As without first using Reset, no longer deletes an MCD modified relational work offset.

A new macro, LockAxisOnOff, is added to enable locking a particular machine axis during simulation. An error message is output when a locked axis is moved.

Support is added for setting a variable to empty (null) using VERICUT macros by setting the Override Value to #0.

Miscellaneous

Using OpenGL with VERICUT is no longer supported on UNIX computers.

Unexpected VERICUT termination no longer occurs when using the IGES Converter with an extremely long Input file name.

The environment variable UGII_ROOT_DIR, is added to the path environment variable in the vericut.bat file for both 32 and 64 bit Windows.

A new math operator, "Div" is added to "Math Sub Type" on the Word Format table, to support the Siemens 840D DIV operator.

Div:

The Div mathematical operator divides the first number (value1) by the second number (value2), truncates the results, and assigns the results to the specified word, where word can be either a variable or a machine component.

The expected syntax is:

Word = value1 DIV value2

For example:

Word = 10 DIV 3 = 3

Word = -8 DIV 3 = -2

#1 = 12 DIV 3 = 4

Optimizing with the Library tool change sub file, "maz640mpro.sub", in use no longer causes blocks from the subroutine to be output to optimized file.

The Library hei530.ctl is updated to no longer use the CUTCOM state stored in the R0 WORD.

The Library control, mazatrol_matrix_m.ctl, is updated to support an R on a cycle motion block when a cycle G-Code is not present.

Encrypted files now work correctly with V7.0 Limited licenses.

The "LSERVOPTS" environment variable can now be created in the "system environment" for both VISTA 32 and Vist64 computers.

Unexpected VERICUT termination no longer occurs when using Model Export to create an IGES file for a specific VERICUT project file.

Tool snapshots used in VERICUT reports are now captured correctly for tools that do not have a cutting portion.

In the File menu > Preferences: Startup tab, after clicking on the color pallet icon to specify the Mouse Pick Highlight Color, the color pallet that displays can now be dismissed without selecting a color by clicking on the "x" in the upper right corner of the color pallet.

The dimensions in a Setup Plan report are no longer affected by the machine position. Dimensions are output relative to the Stock component.

The Setup Plan view is now based on the "active" (highlighted) VERICUT view.

All Sample and Library file names and extensions have been set to lower case to prevent problems when used on UNIX computers.

The huron_kx100_sin840d.vcproject file located in the \samples\huron\ directory of your V7.0.1 VERICUT installation and in the VERICUT Samples download area accessed from the CGTech website is enhanced. The following is a summary of the enhancements:

- Added a CYCLE978 Siemens probing cycle to define G54 values.
- Added a facing operation.
- Added a Tap operation with error.
- Modified the cavity milling with one less pass to have simulation go faster

All V7.0 Training sessions are now included in the VERICUT Training Session section of the V7.0 CGTech Help Library.

Training files required by the V7.0 Training Sessions are now included in the /training/directory of your V7.0.1 VERICUT installation.

The behavior of NC Program Review is now consistent when more than one tool is milling at the same time (using the SetVcMultiTools that allows all loaded tools to mill at the same time).

Machine Time calculations are now correct for situations where no Stock model exists.

Adding or removing the last icon in the tool bar no longer cause bad behavior when the Display Toolbar check-box is checked in the View Toolbar dialog.

The path of the tool, represented by colored lines, in NC Program Preview views are now displayed correctly in a Machine/CutStock view.

When simulating a wire EDM operation, the wire is now displayed correctly in Shaded draw mode. In addition, when the wire is cut, the display is updated simultaneously in all views.

Machine components in a specific project file no longer disappear from the display when Zooming in NC Program Review.

The section plane no longer disappears when "Refine Display" is invoked while X-Caliper and "Display Holder Collisions" are both active.

After using the Tool/Stock Collisions feature in X-Caliper, the X-Caliper window no longer needs to be open in order for Refine Display to "reset" to the display to normal.

The report template file name is now displayed in the report template window's title bar regardless of whether the file is directly opened in the report template dialog or if it is opened because of being referenced in the .vcproject file.

CATIA V5-to-VERICUT Interface (CATV5)

CATV5 no longer creates an axis system for un-named axis systems in CATIA.

Left Hand External Turning Tools and Right Hand Internal Turning Tools that are defined and work properly in CATIA, are no longer inverted when passed through CATV5 to VERICUT.

EdgeCAM-to-VERICUT Interface

The Edgecam-to-VERICUT Interface now correctly loads machine models for a specific Mill/Turn machine when using Edgecam 11.

The Edgecam-to-VERICUT Interface now correctly loads all generated Stock models in VERICUT when using Edgecam 11.

The Edgecam-to-VERICUT Interface now correctly loads the STL machine models it creates in VERICUT when using Edgecam 11.

The Edgecam-to-VERICUT Interface now correctly transfers tool holders when using Edgecam 11.75.

Unexpected Edgecam-to-VERICUT Interface termination no longer occurs when using solid fixture models and Edgecam 11.75.

The Edgecam-to-VERICUT Interface now correctly transfers and loads stock models created from a solid model when using Edgecam 11.75.

The Edgecam-to-VERICUT Interface now correctly transfers and loads cylindrical fixture models when using Edgecam 11.75.

The Edgecam-to-VERICUT Interface no longer requires you to check "Active" on the VERICUT Setup Interface: General tab in order to transfer the selected sequence, when only 1 sequence is present in the Part.

The Edgecam-to-VERICUT Interface is enhanced to reduce the time that it takes to launch VERICUT.

Unexpected EdgeCAM termination no longer occurs when you generate CNC code after using the Edgecam-to-VERICUT Interface. Edgecam fixed this problem in Edgecam2009 R1.

"Vericut.commands" file required by the Edgecam-to-VERICUT Interface is now included in the VERICUT installation.

The Edgecam-to-VERICUT Interface is enhanced to enable using a Tool List to do tool changes by alphanumeric Tool Descriptions rather than by Tool ID's.

The Edgcam-to-VERICUT Interface is enhanced to enable the transfer of user-defined cutters.

GibbsCam-to-VERICUT Interface (GibbsV)

GibbsV is enhanced so that if the "Tool ID #" option in the GibbsCAM tool definition window is checked, then the TOOL ID that is specified in the text field is transferred to VERICUT. If "Tool ID #" option in the GibbsCAM tool definition window is NOT checked, then the tool's location in the tool table (Tool sequence number) is transferred to VERICUT.

MasterCAM-to-VERICUT Interface (MCAMV)

A "Stock Csys." feature is added to the MCAMV main window. This feature is similar to the "Model Csys." feature except that it only applies to the Stock model. This feature enables you to specify how the CutStock transfers from one setup to the next.

MCAMV is enhanced to keep the Stock Component on the Fixture Component, when no Fixture models are present, during the transfer of the Stock Component from one setup to the next.

MCAMV now transfers lathe tools correctly to VERICUT.

NX-to-VERICUT Interface (NXV)

NXV no longer outputs an error message after selecting the Add button to add NC Sub Program files.

Pro/ENGINEER-to-VERICUT Interface (Proev)

A new Pro/ENGINEER-to-VERICUT Interface is now available.

New Macros in V7.0.1

The following new macros are added for V7.0.1.

AbSyncChannelCommand
AdjustToolOffset
CycleFeedCheckOnOff
LockAxisOnOff
OptiEOLStr
Siemens840DGoto
TapeCourseTValue
TapeMTorresCourseID
TapeMTorresTowInv
VirtualZAxisABDynamic
VirtualZAxisABRotary

Macros not yet included in the documentation

AbSyncChannelCommand

Function — SYNC

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — Value

This is an Allen-Bradley specific macro. The Allen Bradley documentation states that:

"Start Of Block synchronization occurs when a block of information contains..."

"Any command, or combination of commands, that affects both axis pairs".

This macro is used to track any of these commands that are present on the current block. If a block contains one of these commands for both channels, then a sync will be done.

The value passed to this macro represents the channel that this command is associated with.

1 = command for channel 1

2 = command for channel 2

3 = command for both channels.

AdjustToolOffset

Function — TOOL OFFSETS

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — Value

This macros allows flexibility on how the tool offsets are adjusted. Typically, this is always set to be along the Tool's coordinate system.

NOTE: This is the only option that will work well with RTCP, RPCP, or dynamic work offsets.

We now also support not adjusting the tool offset, and adjusting the tool offset to be along the local coordinate system.

The value determines how the tool offsets will be adjusted:

0 = Do not adjust the tool offsets

1 = Adjust the tool offsets to be along the Tool coordinate system (Default)

2 = Adjust the tool offsets to be along the local coordinate system

NOTE: This macro does not cause the tool offsets to be recalculated, it only sets a flag which will determine how they are adjusted when they are recalculated.

AxisMotion

Function — MOTION

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

Text = the axis to be moved.

Sets the axis to be moved ("A", "B", "C", "A2", "B2", "C2", "U", "V", "W", "X", "Y", "Z") then calls the appropriate axis motion macro passing along the Override Text value. For example, if Override Text = B, then macro BAxisMotion is called.

NOTE: Macro AxisMotion should not be used when you need to specify the override text "-" to select the motion direction when in the "absolute" mode.

Also see: AAxisMotion, BAxisMotion, CAxisMotion, A2AxisMotion, B2AxisMotion, C2AxisMotion, UAxisMotion, VAxisMotion, WAxisMotion, XAxisMotion, YAxisMotion and ZAxisMotion

AxisMachineMotion

Function — MOTION

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

Text = the axis to be moved.

Similar to AxisMotion except that it sets the axis to be moved ("A", "B", "C", "A2", "B2", "C2", "U", "V", "W", "X", "Y", "Z") then calls the appropriate axis machine motion macro, passing along the Override Text value. For example, if Override Text = B, then macro BAxisMachineMotion is called.

Also see: AAxisMachineMotion, BAxisMachineMotion, CAxisMachineMotion, A2AxisMotion, B2AxisMachineMotion, C2AxisMachineMotion, UAxisMachineMotion, VAxisMachineMotion, WAxisMachineMotion, XAxisMachineMotion, YAxisMachineMotion and ZAxisMachineMotion

CheckVolumeAboveDepthLimit

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

Value = the maximum allowable volume of material that can be removed by the portion of the cutter above the Maximum Cut Depth
positive value = enables this feature
negative value or 0 = disable this feature

When this macro is called, a “special” volume removed by each cut is calculated by VERICUT. This “special” volume is based on the material removed by the portion of the cutter above the Maximum Cut Depth. When the “special” volume is greater than or equal to the specified value, a Warning message is output to the Logger. Enter a positive value, representing the maximum allowable volume in the Override Value field, to enable this feature. Entering a negative value, or 0, in the Override Value field disables this feature.

This feature only works for “standard” VERICUT (not FastMill). If this feature is enabled when FastMill is active, VERICUT will internally turn off FastMill mode.

If OptiPath is turned on while this feature is enabled, the OptiPath results will be based only on the material removed by the portion of the cutter below the Maximum Cut Depth.

Use of this macro requires the following:

1. **Check Cutting Limits** in the Project Tree, Setup Configure menu: Motion tab must be toggled “on”.
2. **Maximum Cut Depth** on the OptiPath window: Limits tab must be enabled.

CircleArcAngle

Function — MOTION

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

Value = the arc angle specified as a full angle of rotation

Use to enable specifying a helical/circular motion by a full angle of rotation. For example, the Heidenhain 530 control uses the “B5=” field to specify the arc angle as shown below:

```
G01 Z9. F100  
X30 Y30 F500  
G2 I40 J20 K-8 B5=420
```

CollisionPreCheckOnOff

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

0 = Off (default)

1 = On

If the collision pre-check option is turned on, we will first do the collision check using an automatically created simplified primitive. If this check indicates that there might be a collision, the collision check will be done with the original primitives.

Currently the default for the collision pre check is Off. Theoretically, if the pre check always causes the collision check to be done with the original primitives, then it would be faster not doing the pre-check. If collision checking performance is slow on the job that you are working on, you can try turning the pre-check on, and see if this helps the performance.

CompositeValue

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text, Value

Text = composite format : variables to which the individual components are assigned

Value = the composite value

Use this macro to breakdown a composite value into its individual components, and store each component in the variable specified. The text value contains the composite format (for example: 2 3*.2), followed by a ':', followed by a series of variable names (up to 10) indicating where to store the individual components. All arguments are separated by spaces.

If the breakdown of the composite value does not result in an individual component for a specific variable, then that variable will be set to zero as shown in EXAMPLE 2 below.

EXAMPLE 1:

Override Value = 1234567.89

Override Text = 2 3*.2 : VAR1 VAR2 VAR3 VAR4 (Note the space between the 2 and the 3)

2 3*.2 : VAR1 VAR2 VAR3 VAR4 is interpreted as:

Put the first 2 characters of the composite value (12) in VAR1.

Put the next 3 characters of the composite value (345) in VAR2.

Put all remaining characters up to the decimal point (67) in VAR3.

Put the characters after the decimal point, in this case up to 2 (89) in VAR4.

The variables would be set as follows:

VAR1 = 12

VAR2 = 345

VAR3 = 67

VAR4 = 89

EXAMPLE 2:

Override Value = 12.1

Override Text = 2 3*.2 : VAR1 VAR2 VAR3 VAR4

2 3*.2 : VAR1 VAR2 VAR3 VAR4 is interpreted as:

Put the first 2 characters of the composite value (12) in VAR1.

Put the next 3 characters of the composite value (none) in VAR2.

Put all remaining characters up to the decimal point (none) in VAR3.

Put the characters after the decimal point, in this case up to 2 (1) in VAR4.

The variables would be set as follows:

VAR1 = 12

VAR2 = 0

VAR3 = 0

VAR4 = 1

ConvertArrayIndex

Function — EVENTS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

0 = do not apply the AXIS variable conversion

1 = apply the AXIS variable conversion to index (default)

This macro is used to enable/disable the conversion of the array index, specified as AXIS variable, or the constant in the argument string specified with the **SetDynamicVars** macro. If this macro is not used, or called with value of 1, all array references in argument string are decoded when the AXIS variable/constant is used. If a value of 0 is specified, the old parsing logic is used where the array element can only be referenced by numerical index.

Also see: **SetDynamicVars**

CutterCompOnOffSwitches

Function — CUTTER COMPENSATION

Status — ACTIVE

Comment — Added V6.2.2

Valid Inputs — Text, Value

This macro is used to set the flags which determine how to process the turning on and turning off of Cutter Compensation.

Value: Specifies the option you want for a specific scenario as defined by the Text argument. The valid options are specific to the specific scenario selected. Valid Options are:

Text = 1

Value = 1 → Ramp On

Value = 2 → Immediate

Text = 2

Value = 1 → Ramp On with Active Plane Motion

Value = 2 → Immediate

Value = 3 → Ramp on with Any Motion

Text = 3

Value = 1 → Ramp On with Active Plane Motion

Value = 2 → Immediate

Value = 3 → Ramp On with Any Motion

Text = 11

Value = 1 → Adjust the input coordinates to match the physical location of the tool.

Value = 2 → Do not Adjust the input coordinates to match the physical location of the tool

Text: Specifies which flag (scenario) to set:

1 = Turning on Cutter Compensation with motion within the plane

2 = Turning on Cutter Compensation with no motion

3 = Turning on Cutter Compensation with the only motion being outside of the plane

11 = Turning off Cutter Compensation

CycleFeedCheckOnOff

Function — DRILL CYCLES

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — Value

0 = turn error message Off

1 = turn error message On (default)

Internally VERICUT converts the cycle parameters into distance to "RAPID TO" and distance to "FEED TO", measured from the part surface, and measured along the Tool's Z axis. If the "FEED TO" value is larger than the "RAPID TO" value, and this is not a back boring operation, then VERICUT reports an error. This macro is used to turn this error message on and off.

A value of zero turns the error message "off", and a value of 1 turns the error message "on". The default is on.

NOTE: This error should only be turned off in very rare situations.

CycleTurnDiameter

Function — TURNING CYCLES

Status — ACTIVE

Comment — Added V6.2.2

Valid Inputs — None

Macro **CycleTurnDiameter** sets the turning cycle type to diameter. The current position is used as the diameter start point. When X and/or Z words are included on the block they define the endpoint of the canned diameter. When U and/or W words are included on the block, the associated U-W values are interpreted as signed incremental values from the start point to the end point. The **XaxisIncreMotion** and **ZaxisIncreMotion** macros must be called when U and W are used to define the canned diameter endpoint.

The **CycleTurnDiameter** macro is modal until canceled by one of the following macros: **MotionRapid**, **MotionLinear**, **MotionCW**, **MotionCCW**, **MotionNurbs**, **MotionPoly**, or **Motion3DCircle**.

The G90, canned diameter cycle remains active allowing additional blocks of the G90, cycle to be simulated until canceled by another G-code.

A G90, word address is required to call the **CycleTurnDiameter** macro. The associated **CycleTurn*** macros ***DO NOT*** have to be called for each block.

For example:

```
G90 X2.5Z-2R0.1  
X2.4  
X2.3  
X2.2  
X2.1  
X2.0  
G00X3.0Z1.0
```

CycleTurnTaper

Function — TURNING CYCLES

Status — ACTIVE

Comment — Added V6.2.2

Valid Inputs — Value

Specifies the distance of taper in X-axis direction for a diameter canned cycle (signed value, radius).

CycleTurnThreadOff

Function — TURNING CYCLES

Status — ACTIVE

Comment — Added V6.2.2

Valid Inputs — Text, Value

The macro enables you to define a set of word/value pairs in the OVERRIDE TEXT field which are used to signify when the turning cycle modal is turned off. For example, the word/value pairs entered in the OVERRIDE TEXT field might be as follows: "G00, G01, G02, G03". This provides full control for cancelling turning cycle modals.

CycleTurnThreadStartAngle

Function — TURNING CYCLES

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

Value = the thread starting orientation

This macro sets the thread starting orientation (0 to 359.999 degrees) for multiple threads in a general thread turning cycle.

DebugComponentAxis

Function — MISCELLANEOUS

Status — SPECIAL

Comment — Added V6.2.2 - This macro is intended for internal CGTech use only.

Valid Inputs — Text

This macro prints the location of the specified component.

GlobalVariables

Function — VARIABLES

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

This macro defines the ranges for global variables. NOTE: This macro only applies to positive integer variables.

The format for the TEXT value is:

-30, 50-70, 90, 300-400, 500-

The above line would define the following variables as global:

- All variables 30 and below.
- 50 through 70
- 90
- 300 through 400
- All variables 500 and above

This macro now makes SetMaxSubsystemVar and SetMinSubsystemVar obsolete.

LockAxisOnOff

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — Text, Value

Text:

The name of the machine axis ("A", "B", "C", "X", "Y", "Z", "U", "V", "W", "A2", "B2", "C2")

Value:

- 0 = Lock the machine axis
- 1 = Unlock the machine axis

This macro is used to lock, or unlock a particular machine axis. If the NC program moves a locked machine axis, an error message is output.

MatchParensInComment

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

- 0 = a comment defined by parentheses will close on first ')' (default)
- 1 = a comment will not be closed unless *all* opening parentheses "(" are matched by closing parentheses ")"

This macro is used to specify how VERICUT should determine when the end of a comment is reached. It should be called at the start of processing.

If "(" is defined as a "Begin Comment" word in the Word Format window: the comment will end on the first ")" if the Override Value = 0, or the matching ")" if the Override Value = 1; however the comment will always end on End of Block.

EXAMPLE 1

"N0004 (THIS COMMENT (1.000) HAS MATCHED PARENTHESES)"

the comment will end on the first ")" if the Override Value = 0),

"N0004 (THIS COMMENT (1.000))"

or by the second ")" if the Override Value = 1.

"N0004 (THIS COMMENT (1.000) HAS MATCHED PARENTHESES)"

EXAMPLE 2

"N0004 (THIS (COMMENT (1.000) HAS UNMATCHED PARENTHESES) SO WHAT HAPPENS?"

the comment will end on the first ")" if the Override Value = 0,

"N0004 (THIS (COMMENT (1.000))"

End of Block if the Override Value =1.

"N0004 (THIS (COMMENT (1.000) HAS UNMATCHED PARENTHESES) SO WHAT"

If "(" is defined as a "Multiline Comment" word in the Word Format window, the End of Block is irrelevant:

EXAMPLE 3

"N0004 (THIS (COMMENT (1.000) HAS UNMATCHED PARENTHESES) SO WHAT HAPPENS?)"

the comment will end on the first ")" if the Override Value = 0,

"N0004 (THIS (COMMENT (1.000))"

or on the matching ")" if the Override Value = 1.

"N0004 (THIS (COMMENT (1.000) HAS UNMATCHED PARENTHESES) SO WHAT HAPPENS?)"

McallSubroutineOn

Function — SUBROUTINES

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text, Value

This macro is used to save the subroutine name and retain the parameters upon returning from the subroutine. McallSubroutineOn should be called when a Siemens 840D **MCALL** *subroutine name* command is encountered. It should be followed by a call to the **Siemens840DProcCall** macro. An **MCALL** command on a block by itself should call the **CycleSubroutineOff** macro.

How it works:

MACALL *subroutine name* passes the subroutine name and parameters to the McallSubroutineOn macro which saves the subroutine name and parameters so that they are available upon returning from the subroutine. Siemens840DProcCall macro invokes the call to the subroutine. When an MCALL command is the only thing on a block, the **CycleSubroutineOff** macro is called to cancel the above actions.

EXAMPLE:

```
X50 Y0 Z55  
MCALL CYCLSUB(60,50,2,35)  
Y30  
MCALL
```

In the above example, the CYCLSUB(60,50,2,35) subroutine is called twice before the MCALL block cancels the above sequence.

MSCloseFile

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — None

This macro is designed to support commands similar to the Fanuc PCLOS command. This macro closes the external text file opened using macro MSOpenFile. This macro should be called after processing the last MSWriteToFile command, or before the end of the NC program. Macros MSOpenFile and MSCloseFile should always be used in pairs.

MSOpenFile

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

This macro opens an external text file for the output provided by the MSWriteToFile macro. The Override Text field is used to specify the name of the output file. If the file name is specified without a path, it will be located in the VERICUT Working Directory. If the specified file exists, the formatted string output by MSWriteToFile is appended to its contents. This macro must be used before any MSWriteToFile macro is processed. Note that only one external file can be opened at any time. The file should be closed, using macro MSCloseFile, after the last MSWriteToFile macro, or before the end of the NC program. Macros MSOpenFile and MSCloseFile should always be used in pairs.

MSWriteToFile

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — None

This macro is used to format, and output, NC program variables similar to the Fanuc DPRNT command. The only input data for this macro is the text following the DPRNT command. The variables and their format are specified in DPRNT statement. See Fanuc 15i programming manual page 659 – 663 for details. The Fanuc system variable #7000 (bit 7) can be used to control the current format design (see Fanuc manual for details). The formatted string is output to an external file opened using macro MSOpenFile.

OptiEOLStr

Function — OPTIPATH

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — Text

This macro is used to define a string that should be at the end of every line of an optimized output file (except for empty lines). When optimizing a file, the OptiPath will insure that every optimized line output will contain this string.

NOTE: Lines that are not optimized, will be passed directly from the input to the output file.

The default EOL (End of Line) string is the empty string (nothing).

RelationalOffsetRegisterName

Function — TOOL_OFFSETS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

This macro has been created as an alternative way to specify the component in which a relational offset should be applied. Rather than calling one of the RelationalOffsetCompName macros, this macro can be called to specify the Register name that you want the offset to apply to. For example: If your machine is defined with both a X and a U register component, both moving along the X axis, if you call this macro with an OT=U, then the X relational offset will be applied to the U register component.

If a text value of “RESET” is specified, the direct specification of the registers used for X, Y, Z Relational offsets will be deleted, and the software will apply X to X, Y to Y and Z to Z.

RestoreMotionType

Function — MOTION
Status — ACTIVE
Comment — Added V7.0
Valid Inputs — None

This macro restores the motion type to the type of the last non-rapid motion. This macro is only applicable when in RAPID mode, and you want to switch back to the previous non-RAPID mode. If the current motion type is set to something other than RAPID, this macro does not restore a previous mode, and will keep the current motion type.

RestorePrevMotionType

Function — MOTION
Status — ACTIVE
Comment — Added V7.0
Valid Inputs — None

This macro is used to restore the previous motion type.

NOTE: Commands like ToolChange, CyclesExecute and CyclesCancel may change the motion type.

RestoreSavedMotionType

Function — MOTION
Status — ACTIVE
Comment — Added V7.0
Valid Inputs — None

This macro restores the "saved motion type". The "saved motion type" is set by calling the **SetSavedMotionType** macro.

RollerFormCollisionIgnore

Function— TAPE LAYING
Status — ACTIVE
Comment — Added V7.0
Valid Inputs — Value

0 = turn off the ignore flag
1 = turn on the ignore flag

This macro enables, and disables, the normal collision checking that would occur between the active tool (Roller) and the active stock (Form). It does this by modifying the ignore flag on the corresponding entry in the collision table.

NOTE: The corresponding entry must already exist in the collision table.

A value of 0 turns off the ignore flag, and a value of 1 turns on the ignore flag.

NOTE: The ignore flag was implemented in V7.0. It is being back ported into V6.2.x to support this command, but without the corresponding GUI update to the collision table.

RotationPlaneXMirror

RotationPlaneYMirror

RotationPlaneZMirror

Function — MIRROR

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — none

These macros are used to add mirroring support to Siemens 840D \$P_PFRAME. These macros are equivalent to the **MirrorX**, **MirrorY** and **MirrorZ** macros where the mirror axis value is 0. When one of these macros is called, the corresponding axis will be reversed. Circular interpolation is not supported when mirror is activated by one of these macros.

SetCycleFeedrate

Function — DRILL CYCLES

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

This macro sets the feedrate which is used during a drill cycle while moving from the Rapid plane to the final Depth level. This Feedrate is independent of the normal feedrate used when not executing a drill cycle. The default value is zero, which means to use the normal feedrate. This value is modal. This means, it might need to be reset to zero when the drill cycle is defined (G81, G82, ...).

SetCycleFeedrate2

Function — DRILL CYCLES

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

Sets the secondary feedrate which is used during specific drill cycles while moving from the Final Depth level back to the Rapid plane. This Feedrate is independent of the normal feedrate used when not executing a drill cycle. The default value is zero, which means to use the main cycle feedrate (See SetCycleFeedrate). This value is modal. This means, it might need to be reset to zero when the drill cycle is defined (G81, G82, ...).

NOTE: This feedrate only applies to BORE, BORE DRAG, and TAP cycles.

SetSavedMotionType

Function — MOTION

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — None

This macro is used to set the "saved motion type" which is used by the RestoreSavedMotionType macro.

Siemens840DGoto

Function — Branching

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — Text, Value

Text:

The Label to be processed

Value:

-1 = search backwards

1 = search forwards

0 = is search forwards and then backwards if not found.

This macro is for Siemens 840D Goto command. If the text argument is a 'N' followed by a number, then a goto sequence will be processed. Otherwise a goto label will be processed. The value determines the direction of the search:

NOTE: Since the text argument defines the label or sequence number, no other macro calls are needed to set label or sequence number to branch to.

NOTE: The GOTO, GOTOF, GOTOB, GOTOC words needs to be defined as having an alpha-numeric argument.

TapeCourseTValue**Function** — TAPE LAYING**Status** — ACTIVE**Comment** — Added V7.0.1**Valid Inputs** — Value

This macro is used to define the course T Value, a parametric parameter that states how far along the course we are. This is typically used with the pre-definition of when tows are to be turned on and off.

TapeMTorresCourseID**Function** — TAPE LAYING**Status** — ACTIVE**Comment** — Added V7.0.1**Valid Inputs** — Value

This macro is used to define the course ID. This macro is specific to MTorres because it is also being used to match up with the "tape_id" field in the corresponding Tow File.

TapeMTorresTowInv**Function** — TAPE LAYING**Status** — ACTIVE**Comment** — Added V7.0.1**Valid Inputs** — Value

By default on the MTorres machine, Tow 1 is the right most tow when looking behind the head in the direction of motion. This macro, when passed a value of 1, reverses the order so that Tow 1 is the left most tow.

NOTE: When displaying Tow statistics, Tow 1 is always the left most tow. This macro has no effect on this display. This macro is strictly used to interpret the input.

TravLimitErrorReporting**Function** — MISCELLANEOUS**Status** — ACTIVE**Comment** — Added V7.0**Valid Inputs** — Value

Value = error reporting mode

Use to define the travel limit error reporting mode. An override value of "0" produces error reporting for all motion (default). An override value of "1" prohibits error reporting during a rapid motion.

ViperCourseDist

Function — TAPE LAYING

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

This macro is specific to the Cincinnati Viper Tape machine. It defines the instantaneous distance for the current course (since starting to lay this piece of tape).

NOTE: Although this appears to be a fairly generic value, internally, this value is important to various calculations specific to Viper.

ViperTowCut

Function — TAPE LAYING

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

This macro is specific to the Cincinnati Viper Tape machine. It defines the tows that are currently being cut, or has been cut. Tows that are currently being cut will run out in distance specified by the **ViperTowPrefeedDist** macro.

For example: T2063

Each bit is associated with a tow. 2063 converted into base 16 is: 80F This value corresponds to 4 tows on the low side, and 1 tow on the high side.

ViperTowPinch

Function — TAPE LAYING

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

This macro is specific to the Cincinnati Viper Tape machine. It defines the tows that are to be restarted. The tow will actually start being laid in the distance specified with the **ViperTowPrefeedDist** macro.

For example: Sa2048

Each bit is associated with a tow. 2048 converted into base 16 is: 800. This value corresponds to tow 12.

ViperTowPrefeed

Function — TAPE LAYING

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

This macro is specific to the Cincinnati Viper Tape machine. It defines the distance that a tow should be pre-fed prior to the "headon" command. Subtracting the specified value from the **ViperTowPrefeedDist** will give you the distance the head will need to travel before the tow actually starts laying tape.

The specific tow is referenced by A-M, O-Z, and a-g

Example: Text value = "A2.9B2.9C.5D2.5E2"

ViperTowPrefeedDist

Function — TAPE LAYING

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Value

This macro is specific to the Cincinnati Viper Tape machine. It sets the distance from the point where the tape is cut to the point where tape is pressed on to the part. The default is 2.9 inches.

See also: **ViperTowCut**.

VirtualZAxisABDynamic

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — Value

0 = On

1 = Off

Turns On, and Off, the dynamic calling of the VirtualZAxisABRotary macro. If the ABC angles are being calculated from IJK points, then the VirtualZAxisABRotary macro needs to be called after the A and B angles have been set. This is done by calling this macro. A value of 0 turns this feature on, a value of 1 turns this feature off.

VirtualZAxisABRotary

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V7.0.1

Valid Inputs — None

This macro is used to establish a virtual Z axis using rotation planes. The input angle is defined by the local AB axis value. The **AAxisMotion** and **BAxisMotion** macros are used to specify incremental/absolute, and must be called prior to calling this macro. These angles define the angles at which the real Z axis is offset from the virtual orthogonal Z axis.

VirtualZAxisADynamic

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V6.2.2

Valid Inputs — Value

0 = On

1 = Off

This macro turns on, and off, the dynamic calling of the **VirtualZAxisARotary** macro. If the ABC angles are being calculated from IJK points, then the **VirtualZAxisARotary** macro needs to be called after the A angle has been set. This is done by calling this macro. A value of 0 turns this feature on, a value of 1 turns this feature off.

VirtualZAxisARotary

Function — MISCELLANEOUS

Status — ACTIVE

Comment — Added V6.2.2

Valid Inputs — None

Use to establish a virtual Z axis using rotation planes. The input angle is defined by the local A-axis value. The **AAxisMotion** macro is used to specify incremental/absolute, and must be called prior to calling this macro. This angle defines the angle at which the real Z-axis is offset from the virtual orthogonal Z axis.

XAxisMultiplier

Function — MOTION

Status — ACTIVE

Comment — Added V6.2.2

Valid Inputs — Value

Value = Multiplier to be applied to the **XAxisMotion** macro

This macro defines a multiplier that is used by the **XAxisMotion** macro. This multiplier is currently only used by the **XAxisMotion** macro. Typically, this macro will be called when transitioning between programming in radius mode and diameter mode.

If you have a .5 multiplier on the X word or the Word/Address entry, then this macro would be pass a value of 2 when changing to radius mode, and a 1 when changing to diameter mode.

If you do not have a multiplier on the X word or the Word/Address entry, then this macro would be pass a value of 1 when changing to radius mode, and a .5 when changing to diameter mode.

The default value is zero, which means to ignore the multiplier.

NOTE: If a multiplier is defined, and Debug Macro Arguments are turned on, you will see the following results:

Debug: MACRO: XAxisMotion, WORD:X, TEXTSTR=5., VALUE=5

Debug: ADJUSTED X VALUE=2.5 (XAxisMultiplier)

XRelationalOffsetCompName

Function — TOOL OFFSETS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

Specifies the name of the component to which the "X offset" is applied for offsets specified as relational (using Select From/To Locations).

YRelationalOffsetCompName

Function — TOOL OFFSETS

Status — ACTIVE

Comment — Added V7.0

Valid Inputs — Text

Specifies the name of the component to which the "Y offset" is applied for offsets specified as relational (using Select From/To Locations).

ZRelationalOffsetCompName**Function** — TOOL OFFSETS**Status** — ACTIVE**Comment** — Added V7.0**Valid Inputs** — Text

Specifies the name of the component to which the "Z offset" is applied for offsets specified as relational (using Select From/To Locations).

Functions not yet included in the documentation

ISFILE (string)

It is used to check if the specified string refers to an existing file in the system. If file is found the function returns a 1 (TRUE), otherwise the function returns 0 (FALSE). The function checks the current working directory, project directory and library trying to find specified file. The default file extension is MPF. The string argument can contain the file path if necessary.

NOT value

It is used to check if an argument (value) is equal to 0 and returns 1 (TRUE), otherwise returns 0 (FALSE).

VERICUT 7.0.2 Interim Release

VERICUT Version 7.0.2 was released on February 26, 2010. It was a VERICUT Composites Applications only release. This was the initial release of the VERICUT Composites Applications products.

VERICUT 7.0.3 Interim Release

Release Notes

March 31, 2010

VERICUT Version 7.0.3 is available for all supported Windows and UNIX platforms.

V 7.0.3 contains everything described above for V7.0.1, plus the following additional fixes/enhancements.

Verification

The Design component is now transferred between Setups regardless of whether or not it is visible. VERICUT will transfer the Design component according to the following:

- First, VERICUT will try to find a component that is visible in a Workpiece View.
- Next, VERICUT will try to find a component that is visible in a Machine View.
- Finally, VERICUT will try to find a component that could be made visible (i.e., has a model), regardless of current visibility settings.

Unexpected VERICUT termination no longer occurs when a specific CME file created in V6.2 is used in V7.

False "Exceeded Near Miss Tolerance" errors are no longer output for cases when near miss on the cut stock is used and the machine component is oriented fully under the cut stock.

False "Fast Feed Removed Material" error messages are no longer output for situations where an "Exceeded Travel Limits" error is followed by a G1 command.

VERICUT now outputs the following message when Constant Gouge Check is turned "On" and the design component, or design model, is not visible in the Workpiece view.

"The design is not visible in a Workpiece view, thus no collision checking between the tool and the design will occur."

VERICUT now outputs the following message when the Stock component, or Stock model, is not visible in the Workpiece view.

"The cut stock is not visible in a workpiece view, thus no material removal or collision checking between the tool and the fixture or design will occur."

A SYNC code in the middle of a block now processes correctly.

The tool path in a Workpiece view is now correct, for a 90 degree head that uses the Tool Manager: Orientation field to mount the head, for circular moves that are an exact multiple of 180 degrees, or over 360 degrees.

When using Info > File Summary to copy files, the existing directory paths for machine and control files are no longer passed to the copied files.

Unexpected VERICUT termination no longer occurs when Play is pressed to process a specific APT CL file.

VERICUT now correctly detects the collision between a Tool Holder and the sub-spindle chuck in a specific project file.

Unexpected VERICUT termination no longer occurs when processing a G83 cycle in a specific project file.

Material removal is now correct, in a specific project file, when the Animation Speed Slider is set to 100%.

Tool shank and stock collision detection is now consistent regardless of the Animation Speed Slider setting.

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

The Workpiece view display no longer freezes up when using CATIA Dynamic Controls rotating/zooming combination (middle mouse hold, right mouse hold, drag the mouse) when Open GL is active.

The registers for the work offset, and the secondary work offset, are now displayed in the Machine Offsets window.

False "Cycle feed is positive along Tool axis" errors are no longer output for situations where the driven point Z points in the same direction as the tool.

Performing an encrypted "Save As" on machine, or control files, no longer fail when saving the encrypted file is saved to the same directory where the un-encrypted version resides.

Using the Calculate Min Cutter Extension feature no longer alters a specific tool's profile.

New Border Size feature is added, to the Custom Table window and the Picture window for creating Report Template records, to enable defining the thickness of table and picture borders in pixels.

The Stock now displays correctly in the view when a Workpiece view is added.

VERICUT now correctly attaches added Machine and Machine/Cut Stock Views to the Base component instead of to the Stock component.

The No Animation feature now works correctly when the Driven Point Zero axis is displayed.

Motion and other macros are now executed in the order in which they appear in the Word/Address window.

The Project Tree is enhanced so that full /path/filename is displayed in a Tip when the cursor is held over any file name in the Project Tree.

When you highlight a Text variable in the NC-Program window and hold the mouse over the highlighted variable, the value of the variable is displayed in a Tip, just like real/integer variables. You can either left-click, and drag, or double click on the variable name to highlight it.

The helical milling performance is enhanced so that it is now slightly faster than V6.2 instead of slower as it was in V7.0.

An STL model file that has been deleted from a project file and from the project folder, and then replaced with a different STL model file with the same name, now displays correctly in VERICUT.

A specific APT project file created in V6.2 now produces the same results in V7.0.3.

Encrypted controls now work properly with multiple subroutines.

Report Tool Summary via Tool Change by List is now correct for all tools used in a specific project file.

A colon character ":" present in a tool ID no longer prevents saving JPG tool images for use in HTML reports.

Unexpected VERICUT termination no longer occurs when loading a specific project file created in V6.2.

AUTO-DIFF Points model now displays/highlights correctly in a Workpiece view.

The material removal state, for a specific project file, saved in an In Process file is now correct when the In Process file is loaded.

After selecting an NC program in the Project Tree and then right-clicking and selecting "Replace", the file selection box now opens to the directory where the current NC program resides, as it did in V6.2.

When using the 'Step Into Subroutines', 'Step Over Subroutines' or 'Step to End of Subroutine' features, you can now single step by quickly pressing the keyboard space bar.

An In Process file saved from a specific project file, no longer displays an "Error loading in-process file" message when the In Process file is loaded.

Material removal for an ID thread, in a specific SYNC project file, is now correct.

Unexpected VERICUT termination no longer occurs when selecting the Cut Stock in the Project Tree when the Model axis is displayed.

A specific Wire EDM project file created in V6.2 now produces the same results in V7.0.3.

A problem causing some coordinate systems in an NX6 part file to not be transferred to VERICUT, through NXV when the "Transfer All Coordinate Systems" feature toggled "on", is fixed.

The MOVARC/ command, in a specific APT project file, now produces the correct results.

Random unexpected VERICUT termination no longer occurs when a specific project file is processed on 32 bit computers.

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

The simulation when processing an In Process file, that was created when a specific project file was stopped immediately after a tool change command, is now correct.

Copy/Paste of the Cut Stock in the Project Tree from one setup to the next, in a specific project file, no longer "appears" to transfer the Design model when there is insufficient memory to complete the operation. An error message is now output.

In Process files created by "AutoSave" now correctly sets the project file name in the In Process file to the name of the project file that the In Process file was saved from.

The previous tool no longer remains displayed, causing a false collision error, when a turret tool change indexes to an empty tool station.

The "Create CSYS While Simulating" feature now works correctly.

The "Create CSYS From File" feature now works correctly.

NC Program Preview and NC Program Review now display the helical motion, in a specific project file, correctly.

A 3D tool holder, in a specific APT project file, no longer moves differently than the cutter.

Simulation of a specific V6.1.2 project file, with the Animation Speed Slider at 100%, and processing rotary motions with collision checking turned on is no longer slower than it was in V6.1.2.

The G32 threading cycle in a specific project file no longer outputs an invalid fast feed error when the tool exits the thread.

Merging an In Process file, saved at beginning of setup in a specific project file, no longer deletes In Process files downstream from the Project Tree.

Then using Calculate Min. Cutter Extension and Stop At is set to End of each File, VERICUT now prompts you to save the modified tool library at the end of the last NC program file in the current setup.

Processing speed for a specific project file, using the OK to Cut Into Fixture feature, is now comparable to what it was in V6.2.2.

When a new Workpiece view is added, the first Stock component is now automatically used as the 'Attach component'.

The Design model is now positioned correctly when an In Process file, created from a specific project file, is loaded.

Unexpected VERICUT termination no longer occurs, for a specific project file, on a circular motion using an insert cutter profile that has a sharp corner (no radius).

The Design model is now positioned correctly, for a specific project file that uses the automatic transfer of the Design model from one setup to the next, after the project file is saved and then re-loaded.

Unexpected VERICUT termination no longer occurs when single stepping through a specific project file when the MDI window is open.

Unexpected VERICUT termination no longer occurs when saving an In Process file after Setup 1 in a specific project file.

The Cut Stock is now displayed correctly when an In Process file, saved from a specific project file, contains multiple setups with long names and using files with very long directory paths is opened.

Files automatically saved by VERICUT are now written to the correct directory when the "**Auto-set working directory to current project folder**" feature is used.

When a column is inserted into a custom table, for use in a VERICUT Report, now remains in the position where it was inserted after the Table window is closed and then re-opened.

STEP and ACIS models now display correctly in the VERICUT project file when used as machine models.

Material removal during helical cutting motions is now displayed correctly.

When running multiple NC programs in a setup, the program name displayed in the Status window is now correct for each NC program.

Unexpected VERICUT termination no longer occurs when saving a control, or In Process, file that contain invalid FUNCTION_TYPE values.

Replacing an NC program in the Project Tree no longer changes the order of the NC programs.

Cylinders generated by circular motions, in a specific project file, are no longer incorrectly reported by the X-Caliper: Feature History feature as a PLANE.

A specific project file, created in V6.2.1, now produces the same results when processed in V7.0.3.

VERICUT is modified to prevent adding a Stock component under another Stock component, which is an invalid configuration.

A specific project file created in V6.2.2 now produces the same results in V7.0.3.

Probing the stock, when a fixture component is marked as visible and the fixture model is marked as not visible, no longer causes the probe to stop before contacting the stock and report an inaccurate position.

A false "collision between cutter and stock when the tool spindle is off" error is no longer output for a specific project file when a drill is retracting from the drilled hole.

Unexpected VERICUT termination no longer occurs for a specific project file when Pause is pressed.

Text variable support is added for Siemens Sin840D.

Support is added for third parameter, **_ST**, in CYCLE 800 (Swivel Plane) for Siemens Sin840D.

Global User Defined Tags now work correctly when the same User Defined Tag, defined as an empty string, is used in multiple setups.

The tool no longer leaves the contour in the wrong direction (inside the contour) when processing a G62 in a specific project file.

False tool to fixture collisions are no longer reported when the "OK to Cut Into Fixture" feature is used in a specific project file.

Inspection images are now correctly displayed in HTML Inspection Reports.

VERICUT no longer reports false "wrong direction" errors, for the turning spindle direction, when the Animation Speed Slider is set at less than 100%.

Rotary motions are now correct for the Axis Priority settings in a specific project file.

VERICUT-BEGINTABLE and VERICUT-ENDTABLE now correctly creates the table for HTML reports.

The "Holder to Stock Near Miss" feature now works correctly for a specific project file.

The NC program now processes correctly for a specific project file that has two models attached to the Stock component, one of which is invalid.

Machine Simulation

The V axis now moves correctly when a G81 drill cycle is active.

A false collision error is no longer generated for a specific project file.

The CycleMillThread's thread cutting display is now correct when the Animation Speed Slider is set at 100%, or set to Skip Cut.

Helical motion no longer generates false collisions errors when the Animation Speed Slider is set at 100%.

Helical motion no longer generates false "Fast feedrate removed material ..." errors when the Animation Speed Slider is set at 100%.

Support is added for Rexroth Indramotion MTX "CASE-Structure".

Support is added for the Rexroth Indramotion MTX label word "." (the dot) using the **SetPreProcessorType** macro, option 4.

Unexpected VERICUT termination no longer occurs for a specific project file when processing a mill-turn G03 circle block.

Rapid Cycle Time calculations are now correct for situations where 2 rotaries are moving simultaneously and the Max Feed Velocity value is changed from the default value.

False collision errors are no longer generated after touching the part, and then move away from the part using a rotary motion, when probing with a rotary axis.

The cycle positioning move is now correct for a specific project file.

Support is added for Siemens Sin840D CYCLE90 Thread Milling.

The **DynamicToolTipOnOff** macro now works correctly for a specific project file using multiple driven points.

The **DynamicWorkOffsets** macro now works correctly when an additional offset on a rotary axis is active.

New function, ISAXIS, is added to support Siemens Sin840D PROC ISAXIS and the library control file, sin840d.ctl, is configured for its use.

Support is added for Siemens Sin840D PROC VAR AXIS. Axis variables and axis variable arrays are now supported.

Support is added for datum setting using the Siemens command CTRANS().

Support is added for Siemens Sin840D FRAMES.

A specific SYNC project file, created in V6.2.2 and uses mirrored primitives, now processes correctly in V7.0.3.

VERICUT now releases memory correctly at Reset for SYNC jobs loading/unloading detailed tools.

VERICUT now releases memory correctly at Reset for a specific project file using very large Stock and Design models with OpenGL active.

Unexpected VERICUT termination no longer occurs when using Turret Aid to redefine a turret sweep.

Tool Manager

Tool Manager is enhanced to enable transferring Tap tool data from TDM.

The calculations used to determine the default Minor Diameter are modified to eliminate errors for standard tap drill and tap sizes. The default Minor Diameter value is now calculated as follows:

American Unified and National thread:

$$\text{Minor Diameter} = \text{Major Diameter} - (1.299 * \% \text{ full thread} / \text{Threads per Inch})$$

Whitworth:

$$\text{Minor Diameter} = \text{Major Diameter} - (1.281 * \% \text{ full thread} / \text{Threads per Inch})$$

ACME thread:

If Threads per inch ≥ 10 :

$$\text{Minor Diameter} = \text{Major Diameter} - 1 / \text{Threads per Inch} + 0.005$$

If Threads per inch < 10 :

$$\text{Minor Diameter} = \text{Major Diameter} - 0.95 / \text{Threads per Inch}$$

All others:

$$\text{Minor} = \text{Major Diameter} - 1.0 / \text{Threads per Inch}$$

In addition, the Error message text has been changed from "**Error: Tap Stem of ...**" to "**Error: Tap minor diameter of ...**".

Pre-V6.2 Flute length values for "revolved profile" tools are now correctly transformed to the Post -V6.2 method of measuring flute length for "revolved profile" tools.

The cutting (revolved) profile of a milling insert that is not perpendicular to the XP plane is now correct.

VERICUT now utilizes the correct TDM tool list, referenced using the TDM Interface, for multiple setup project files that use unique tool lists for each setup.

Tool Manager no longer loses the connection to the TDM database when a tool library file is saved.

X-Caliper

Measuring a bore using the X-Caliper Feature/History feature now provides the correct Radius and Diameter values when the bore was created using an inserted milling tool.

The Design component is no longer displayed in X-Caliper, Stock Design Distance mode when the Design component is not visible.

The X-Caliper Highlight Same Plane feature now completely highlights the common plane areas.

The X-Caliper Feature/History feature now reports the correct circle information for a specific project file.

The ability to copy "values only" from X-Caliper results is added.

OptiPath

A problem, causing a duplicate line to be output to the OptiPath optimized file when Stop is pressed during the NC block parsing but before the start of the motion processing, is corrected.

Unexpected VERICUT termination no longer occurs after loading a specific IP file and optimizing a specific NC program.

Unexpected VERICUT termination no longer occurs when optimizing a specific extremely large (1GB +) NC program due to running out of memory. A new feature, **Enable NC Program Review**, added to the Properties window: General tab enables turning off NC Program Review. With NC Program Review toggled "off" the amount of memory that VERICUT uses is greatly reduced since it does not need to store the data that is needed by NC Program Review.

A problem causing a poor surface finish, after optimizing a specific NC program that uses helical milling, is corrected.

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

AUTO-DIFF

AUTO-DIFF, Compare by Region now uses less memory when processing very large parts with very small gouge/excess tolerances (0.05mm) on 64 bit computers.

When using AUTO-DIFF Surface method, the design model's position no longer changes after processing when multiple design models reference the same STL file.

The AUTO-DIFF, Constant Gouge/Excess Check, Minimum Desirable Excess setting and value are now stored in the Project file.

Unexpected VERICUT termination no longer occurs when using AUTO-DIFF, Surface method when the first Design model is not visible.

AUTO-DIFF now ignores empty models attached to the design component during AUTO-DIFF processing and the following message, "No visible design model(s) for this stock", is output.

G-Code Processing

Cutter Compensation is now correctly applied for a specific project file that had an active rotation plane.

Unexpected VERICUT termination no longer occurs for a specific project file during the transfer of the turning cut stock from one setup to the next.

Support is added for Siemens 840D commands PAROT and PAROTOF.

Support is added for Siemens 840D system frame \$P_WPFRAME used in CYCLE800 and related to the PAROT command.

You can now get the variable value from a Q word (when selecting the Q variable in the NC code) when the word Q is set to Type: Conditional, Sub Type: HeidCondQWord in the Word Format table.

New macro, **SyntaxErrorOption**, is added to enable controlling the detection and output of error messages during the parsing process of an input mcd code. This feature is designed for specific controls where the NC code is not conventional.

New macro, **SetPreProcessorType**, is added to enable selecting the type of preprocessor that is required to translate NC code to a more standard form accepted by VERICUT.

Some controls, like the MAKA BWO C900, have a very unique NC program syntax and require special conversion of blocks before processing.

Attribute 10, "If the specified Driven Point ID does not exist, use the first driven point if it exists", for the **GageOffsetAttributes** macro now works correctly.

A new Cutter Compensation status is now supported: Cutter Compensation has been turned on within the NC Program, but Cutter Compensation is not being applied. See the table below for details.

	Between Cutter Comp On and Cutter Comp Off in NC Program?	Cutter Compensation being applied by VERICUT?	COMP status light	Cutter Comp text in the Status Window	Text in Cutter Comp State
Status 1 (Same as V7.0.1)	Yes	Yes	Green	ON	ON
Status 2 (New)	Yes	No	Yellow	ON-NO COMP	ON-NO COMP
Status 3 (Same as V7.0.1)	No	No	Dark Green	OFF	OFF

The Status window now displays the correct file name after the first NC program is finished processing.

The machine and local axis variable descriptions are updated to show Machine Axis or Local axis rather than Machine X or Local X if the variable is a non-NUMBER variable (meaning an array or frame variable).

Display of the "Tool Tip Zero" axis no longer causes machine axis motion after Stepping once to create a cut stock.

SYNC now processes correctly for a specific project file with G96 Constant Surface Speed active.

SYNC now produces the correct material removal for situations associated with turning on, or off, the part spindle on channel 2 while cutting in channel 1.

A problem that sometimes caused the NC Program window to be one line out of sync with the status window and VERICUT motion is corrected.

Cycle time calculations are now correct for Siemens 840C L96 roughing cycles.

False Fast Feed errors are no longer output when processing TRICEPT CYCLE83.

TRICEPT error messages now display the correct head and NC program filename.

Cutting time calculations are now correct for a specific turning project where the cutting motion is along a small arc of a very large radius circle.

New macros, **Heid_XAxisIncreMotion** and **HeidYAxisIncreMotion** are added to support Heidenhain RND, CHF, G24, and G25 when motion is incremental.

New macro, **RotaryFeedFactor**, is added to support Okuma's method of feedrate/time calculation with rotaries as mm per minute.

The tool axis vector is now oriented correctly at intermediate points for circular moves that are an exact multiple of 360 degrees.

The GLCondPWord2 now correctly handle variables which contained a zero in it (10, 20, ..., 100, 101, ...).

The **LockAxisOnOff** macro now works correctly when used to unlock an axis, and used to lock an axis, after a tool change.

The correct results are now produced for a specific project file that has two relational work offsets that are re-defined in the NC program and uses a G10 to re-set the first work offset, and then another G10 to set only one axis value of the 2nd work offset.

The ability to create relative work offsets from the NC program (G10L2P...) is added to support advanced features like RPCP, TRAORI, FRAMES and Siemens 840D Cycle 800 that require the work offset table to be relative (ex: Tool to Stock).

When using the Mill Pocket cycle, calling the **CycleMillPocketRefPlane** macro no longer overrides the value set by the **CycleMillPocketRetractPlane** macro.

New Override Text values, "MachineTravelLimits Component_name", "Min_value_varname" and "Max_value_varname" are added to the **SetDynamicVars** macro to enable reading the travel limits of the machine directly from the "machine settings" tables.

The roughing CYCLE95 (VARI= 1 Roughing long. outside) now processes correctly in VERICUT.

VERICUT now correctly identifies a Siemens 840D proc call, if the name of the proc starts with "_" + string (e.g. _ARND) AND a variable is defined also starting with "_" + string (e.g. _AR).

The macro **PolarLinearAxis** is enhanced to support values of -1, -2, and -3. If -1 is specified rather than 1, the polar interpolation will be based on -X values rather than positive X values. Correspondingly, -2 will base polar interpolation on -Y values and -3 will base polar interpolation -Z values.

The **NumCondEqualWord** conditional is enhance to handle WHILE and UNTIL.

New **ExitLoop** macro is added to support a NUM controller's EXIT function.

A 3D circle is now simulated correctly, for a specific project file, where the tool axis is at an angle to the circle plane but the tool axis remains fixed throughout the motion.

The VERICUT library control file, hei530.ctl, is enhanced to have cycle def 7.1 - 7.5 use the **AbsoluteShiftRotationDynamic** macro instead of the **SiemensShiftOffsetA** macro.

SetCyclePartSurface ORV=> -.1 ORT=> 2 is added to the Fanuc master control files.

Library control files, fan3t.ctl and fan3tt.ctl, are now configured correctly for turning. Spindle codes M3, M4, and M5 now call the macro **ActiveSpindleActiveStock** macro instead of the **ActiveSpindleActiveTool** macro.

The library control file, heimplus.ctl, now correctly calls the subroutine, heimplus.sub, when a G7 command is encountered.

The G98 and G99 are now updated in the library control file, mazatrol_matrix_integrex_e.ctl, to use the new drill cycle logic.

The library control file, mazatrol_matrix_integrex_e.ctl, is updated so that:

Registers

Q_1*

(G92) and (R1, R2, or R3)

ActiveSpindleCompName OT:Part_Spindle

ActiveSpindleMinSpeed

S*

(G92) and (R1, R2, or R3)

ActiveSpindleCompName OT:Part_Spindle

ActiveSpindleMaxSpeed

The library control file, phi432.ctl, is enhanced to support drilling cycles.

Changing the session units of a Millimeter project to Inch, no longer causes incorrect cutter compensation values.

The training sample file, process_subroutines.vcproject, now has the machine view correctly attached to Base.

Miscellaneous

The CAD Model Converter no longer converts a millimeter value to an inch value when converting a CATPart to a VERICUT polygon file.

The "Fast Feed Removed Material" error message is now correct when called from non-English VcRes.local files.

The Fanuc library controls are now correctly configured for P dwell with cycles.

The library control file, maz640m_variaxis.ctl, is now configured to use the new drill cycle logic.

Model Export can now export a cut stock that contains tapped holes.

Unexpected VERICUT termination no longer occurs when using **Save Cut Stock > CAD Model, Features Only** method to export a specific Cut Stock in IGES format.

Model Export now recognizes cylinders generated by G2/3 circular motions like it did in earlier releases.

The graphic display now remains correct after opening the Properties window and using the Apply button.

Machine model files are now oriented correctly when reloading a V7 machine file that was encrypted on a 64 bit computer.

File "mcd_matrix_sample.vcproject" is added to the samples folder. The sample demonstrates how (VERICUT-MATRIX i1,i2,i3,d1,j1,j2,j3,d2,k1,k2,k3,d3) is supported in a G-Code file.

VERICUT Reports is enhanced to enable putting a picture/image in a custom table cell.

The library control file, hei530.ctl, is now configured to support the Heidenhain PLANE RELATIVE command.

A Table Header Style feature is added, to enable specifying the colors and fonts used in a table header row different than those used in the rest of the table, for tables used in VERICUT reports.

The ability to use "nested" tables in VERICUT reports is added.

The ability to include a link to an AVI file in a VERICUT report is added.

The ability to record AVI files per tool is added.

The ability to include links to individual AVIs recorded for each tool in a Tool Summary table is added.

Tool and Stock component names are added to the text of "Collision with Spindle off" errors.

Numerous documentation updates, enhancements, and corrections have been completed.

A specific IGES file, created in CATIA, is now correctly imported into VERICUT.

The File Summary, "Copy all files" feature now copies a dll file, specified in the Advanced Control Options: CME/API tab, used in the control file to load macros.

The File Summary, "Copy all files" feature now removes the directory path from a copied dll file.

Helical moves no longer cause long delays when processing a specific project file.

Changing the Fast Feed value during simulation no longer causes unexpected VERICUT termination, or the VERICUT session to freeze up.

CATIA V5-to-VERICUT Interface (CATV5)

CATV now checks the APT file for PPRINT/VERICUT-TOOLID (in addition to LOADTL and TURRET) when determining whether to set the Tool Change By: to "Tool Number" or "Cutter Description".

CATPart models, transferred through CATV, now display correctly in VERICUT.

EdgeCAM-to-VERICUT Interface

The /windows64/edgcam.bat file is no longer empty.

GibbsCam-to-VERICUT Interface (GibbsV)

A number of issues related to passing custom tool holders from GibbsCAM to VERICUT have been resolved.

Solid models in the GibbsCam "Body Bag" are no longer brought over to VERICUT's Fixture, Stock, or Design components.

The only models which reside in the GibbsCam "Body Bag" that are brought over to VERICUT are Tool Holder models referenced by the GibbsCAM Tool Definition window and are only to be used in Tool Manager.

The profile for a specific mill thread tool created in GibbsCAM is now correctly passed through to VERICUT.

The GibbsCAM Tool ID, representing the slot or pocket that the tool resides in, is now correctly passed through to the VERICUT Tool Manager.

GibbsCAM Solid models that include a double quote (") in the model file name can now be successfully passed through to VERICUT. All model file names will have all double quotes (") replaced with an underscore (_) before being output to VERICUT.

GibbsCAM tool descriptions containing the characters "--" can now be successfully passed through to VERICUT. All "--" are now replaced with "-" before being output to the tool library file.

Models with no name in GibbsCam are now successfully passed through to VERICUT.

GibbsV now successfully passes an STL Stock model from GibbsCAM 9.0 through to VERICUT.

Tool Holders from GibbsCAM 9.3.21 are now successfully passed through GibbsV to VERICUT in the correct orientation.

Support is added for reading/parsing encrypted machine files.

Stock, Fixture and Design models are now attached to Stock, Fixture and Design components according to the following:

The first stock, fixture and design models are attached to components named "Stock", "Fixture" and "Design", respectively.

All consecutive stock, fixture and design models will be attached to components named "Stock", "Fixture" and "Design" with a number, starting with the number 2.

For example: "Stock", "Stock 2", "Stock 3", "Stock 4", etc.

GibbsCAM information passed through the GibbsCAM –to-VERICUT Interface now correctly attaches the Machine/Cut Stock View to the Base component instead of to the Stock component.

MasterCAM-to-VERICUT Interface (MCAMV)

MCAMV now launches correctly when the MasterCAM file, or folder path, contain special characters like "ö", or any other accented character.

A new feature, "Put operations into a single VERICUT setup", on the MCAMV Operations window, enables using a MasterCAM Operation for each NC program and then having MCAMV pass the information to VERICUT as a single setup with multiple NC programs.

MCAMV now processes special degree symbol characters correctly enabling the passing of MasterCAM tool descriptions like, <Description>E/M 5/8 X 100° IT#3830</Description>, through to the VERICUT Tool Manager.

MCAMV now outputs the "Program_zero" Csys" and "****STOCK****" Csys correctly in relation to the "Model" Csys.

NX-to-VERICUT Interface (NXV)

The NXV Output Directory is now reset to Teamcenter's temporary folder when NXV is launched with a Teamcenter environment.

NXV is enhanced to populate the Tool Manager Gage Point and Orientation fields when using angled heads within NX. This has been implemented for NX5 and above.

NX counterbore tool profiles are now correctly passed through NXV to VERICUT Tool Manager.

Views changed in one setup are no longer lost if you select another setup when using NXV to transfer data to V7.0.3.

An Assembly Arrangement List feature is added to the NXV Options window. This feature enables you to select from a pull-down list of all NX Assembly Arrangements contained in the current NX part file.

Pro/ENGINEER-to-VERICUT Interface (Proev)

The PROEV license is now correctly released upon exiting from the Pro/E session.

The Default Machining Type is no longer written into the "add vcproject" file when the Project Template is set up for G-Code NC programs.

Support is added for a user preferences file in PROEV.

New Macros in V7.0.3

The following new macros are added for V7.0.3.

AbsoluteShiftRotationDynamicIndex
CaseElse
CaseEnd
CaseStart
CaseValues
ExitLoop
Heid_XAxisIncreMotion
Heid_YAxisIncreMotion
IncrementalShiftRotationDynamicIndex
IVector2
JVector2
KVector2
RotaryFeedFactor
SetLocalVarAccessType
SetPreProcessorType
SiemensPAROT
SyntaxErrorOption

